ADMOULD® 3D-F

CADMOULD® 3D-F FILL

VERSION 6.0

USER MANUAL

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1. Introduction

1.1 Why to simulate an injection molding process?

The complexity of today's plastic parts as well as the costs, quality and competition pressure makes it necessary to recognize potential errors early, already in the development phase of parts and molds in order to solve the problems in the design phase. The most flexible, reliable and economical tool for this is the software CADMOULD® 3D-F. It is available today with sufficient efficiency and is meant to support the experienced injection molding specialist in recognizing and solving problems with the manufacturing process, because simulation is better than trials.

Very often the location of air traps and weld lines is an important topic. With the injection moulding simulation you can optimally define the points of injection or change the wall thickness before the actual injection moulding process in order to shift the location of the weld lines into uncritical areas. You can avoid air traps. The simulation enables you to optimize the process parameters. It quickly supplies you with results for cost reduction of your products.

1.2 What can CADMOULD® 3D-F do?

CADMOULD® 3D-F stands for patented technology (EP 1 385 5103).
The special 3D-framework (3D-F) crosslinking method has the advantage, that the 3D-F mesh, which is needed for simulation, is generated automatically. Please see the appendix for more information.

CADMOULD® 3D-F is a “Computer Aided Engineering (CAE)” software. It is possible to simulate the filling phase of the injection molding process depending on the assigned thermoplastic material and process parameters.

CADMOULD® 3D-F WARP EXPERT users can calculate shrinkage and warpage of the part. As most important results the filling pattern, the pressure loss and the remaining cooling time and warpage are output. Thus different conditions and suppositions can be compared very easily. In addition to the pure filling of the part (re: CADMOULD® 3D-F RAPID) complicate runner systems (hot and cold) and heating/cooling systems can be balanced.
CADMOULD® 3D-F simulates also the cascade injection moulding process. The clamping force limitation and injection pressure capacity of the injection moulding machine may be considered in the calculation of the filling simulation. Based on certain boundary conditions CADMOULD® 3D-F can compute the flow rate profile automatically.
In order to run a simulation, it is necessary to describe the part geometry. Normally the part geometry is available from the CAD system. The CAD format, which is utilized by CAD-

MOULD® 3D-F, is the so called STL-format (stereo lithography - format). This data format can also originate from a 3rd party conversion program.

CADMOULD® 3D-F is also able to simulate multi-cavity and family molds. The connecting runner systems can be defined inside the system or imported from CAD system by IGES files.
Calculations are carried out applying a non-isothermal and, if applicable also compressible, multilayer approach.
The material data base distributed with the program can be edited and extended.

This manual describes the whole function range of CADMOULD® 3D-F.
Depending on the license you have purchased, different functions are available to you.

On this CD-ROM you find also the stand alone program CADMOULD® 3D-F VIEWER:
CADMOULD® 3D-F VIEWER is a program to present all possible results of a CADMOULD® 3D-F calculation.
This software has to be installed separately. To do this, please start the setup in the subfolder "CMViewer" on this CD-ROM.
Please contact your distributor in case of questions about the license.

On the CADMOULD® 3D-F product information CD you find the CADMOULD® 3D-F Viewlets. CADMOULD® 3D-F Viewlets are animated product descriptions, which comes near to a first step tutorial.
The content can also be used for product demonstrations.

1.3 How exact calculates CADMOULD® 3D-F?

By own benchmarking from ***sim**con* in national and international public founded research projects, as well as by CADMOULD® 3D-F users is guaranteed, that CADMOULD® 3D-F RAPID can provide very exact results.
The results of the simulation with reinforced and non-reinforced materials are in the range of the process variation and measuring tolerance.

2. Installation

2.1 Software requirements

Windows® XP/Vista (32bit, 64bit), Windows® 2000, MS-EXCEL® (from Office 97), Browser MS-Internetexplorer or Firefox, Acrobat Reader (V9.0 contained on CD)

2.2 Hardware requirements

fast CPU, option: multi core CPU, min. 1 GB RAM, min 50 GB free hard disk space, graphic adapter with OpenGL support, USB 2.0 port, 3 button mouse, option: SpaceNavigator®, CD-ROM or DVD-ROM

2.3 Before the installation

CADMOULD® 3D-F RAPID can be installed by an administrator only.
CADMOULD® 3D-F RAPID is delivered on CD-ROM. The license file will be send by e-mail.

To install the USB Dongle:

Please, attach the provided dongle to a USB port on your computer. Regard, that the red LED on the dongle lights. In some cases it is possible, that the dongle is not recognized, then please try another USB port.

2.4 Installation

To install please insert the CD ROM into your CD-ROM. The installation procedure should start automatically after a short time. If the installation procedure does not begin, start the routine by hand:

1) Open the Windows-Explorer.
2) Change to the CD-ROM and open the root directory.
3) Start the installation program »Starter(.exe)«

- installation directory
  The directory, where the software is to be installed. The standard is »C:\Program files\Cadmould\Rapid«. For reasons of uniformity the program system should be installed into the given directory.

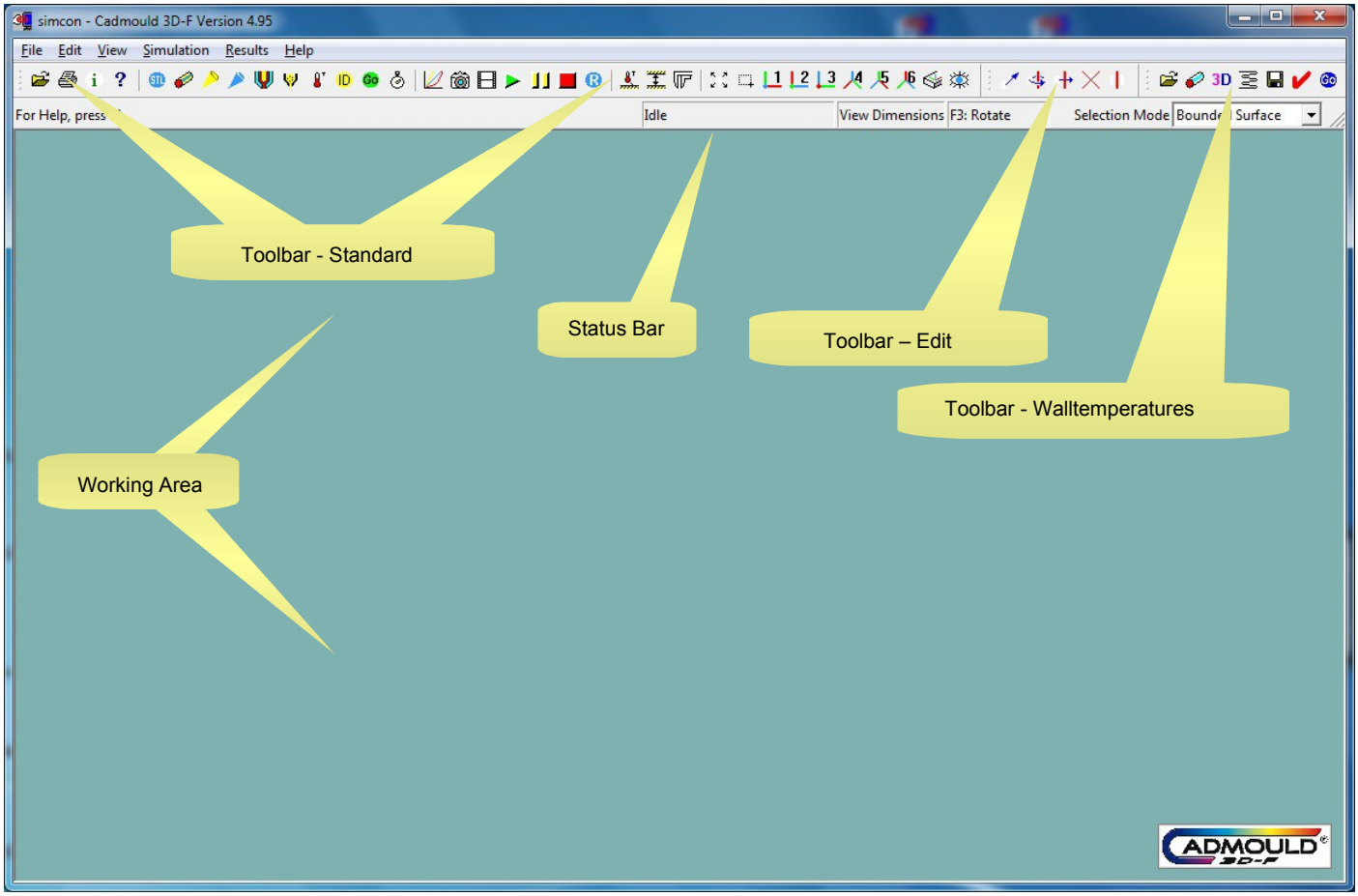
4) Indicate the location of the license file.

- This is the directory where you saved the license file from the mail.

3. Starting CADMOULD® 3D-F

You can start CADMOULD® 3D-F by double-clicking on the icon on your desktop, if this was created during the installation procedure, or you start it by: "Start ⇒ Programs ⇒ Cadmould ⇒ Fill ⇒ CADMOULD® 3D-F ".

Thus you receive the following "start" window:



Please look up specific menus and buttons in the topics "Menu Bar" resp. "Button Bar".

3.1 Command Line Options

The command line options allow to start CADMOULD® 3D-F and CADMOULD® 3D-F Tasklist with user-specific settings.

These options are suffixed to the <cadmould.exe> resp. <tasklist.exe> on the register attributes of the desktop icon. The following options are available among others:

3.1.1 Program language

With the command "-lng language" you can define the language which is used at the start of CADMOULD® 3D-F.

The different languages are abbreviated by the first three characters of the language name.

e.g.: eng; ger; fre; ita; spa; por;

3.1.2 Full Screen

If you add the command "-fsc", CADMOULD® 3D-F is started in full screen mode.

3.2 3D Display with stereo projection

You now have the possibility to display the part stereoscopically.

To use this function you need the following hardware which supports this:

- a graphics card with quad buffering, e.g. nVIDIA Quadro
- shutter glasses
- a display or projector with a frequency of at least 120 Hz

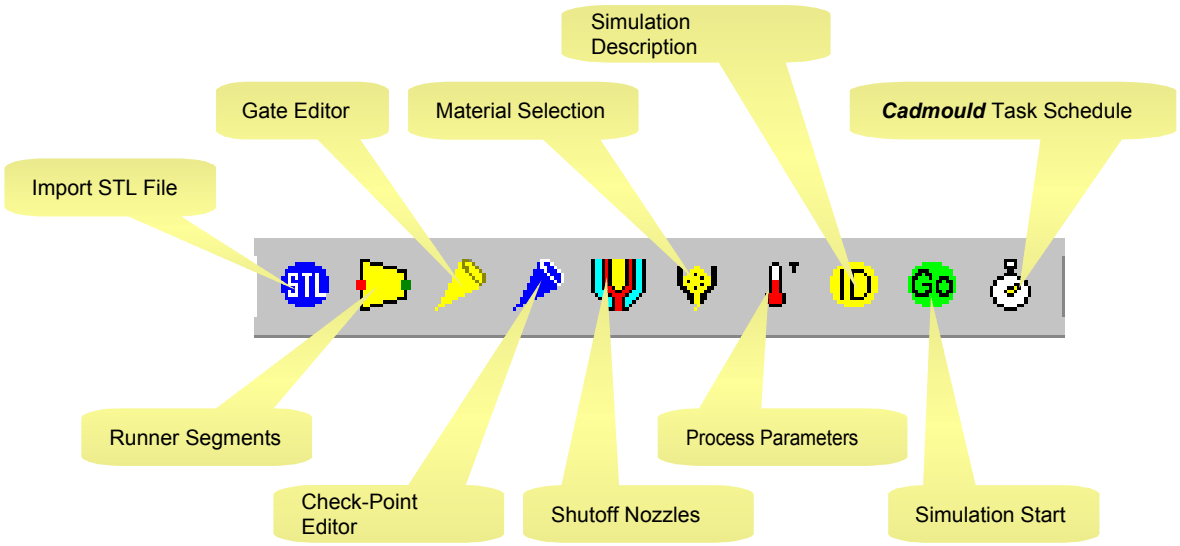
If you want to use this function, you have to start Cadmould 3D-F with the command line option “-prm 2”. You cannot switch between orthogonal and stereoscopic view while the program is running.

Therefore it is advisable to create two desktop icons, one with the command line option “-prm 2” to start the program in stereoscopic mode and one without these option to start it normally.

Please see also chapter: 11.3.13.6 Perspective/Stereo

4. Carrying out a simulation

Perform a simulation by filling in the menus appearing after clicking the buttons from left to the right.



4.1 Load a geometry

You can load a geometry, replace the old one and load a new. Or you can add another (different) geometry to the one loaded. That means, you can merge two different geometries.

4.1.1 Load at the beginning

(*See* chapter*:* Appendix → **STL file**)

(1) Click on the icon "Import STL File".

⇒ The window "Open" appears on screen.

(2) Browse to the directory with the STL-file (stereo lithography file) describing your part geometry. (Example geometries are shipped with the software. You find them in the subdirectory geom in the installation path.)

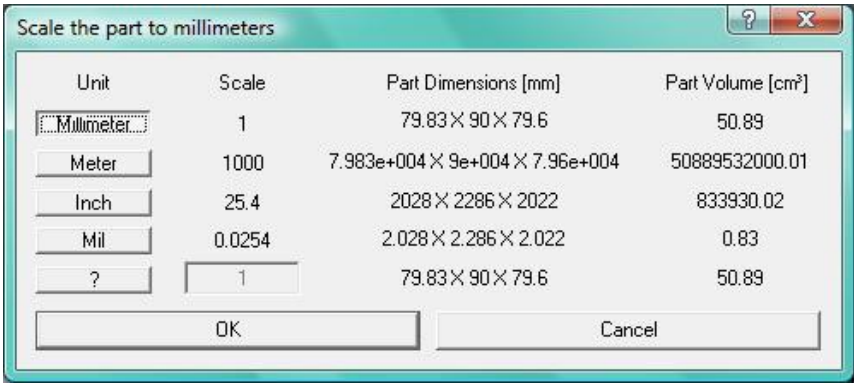
(3) Click on the file: *Example:* becher.stl

(4) Click on "Open".

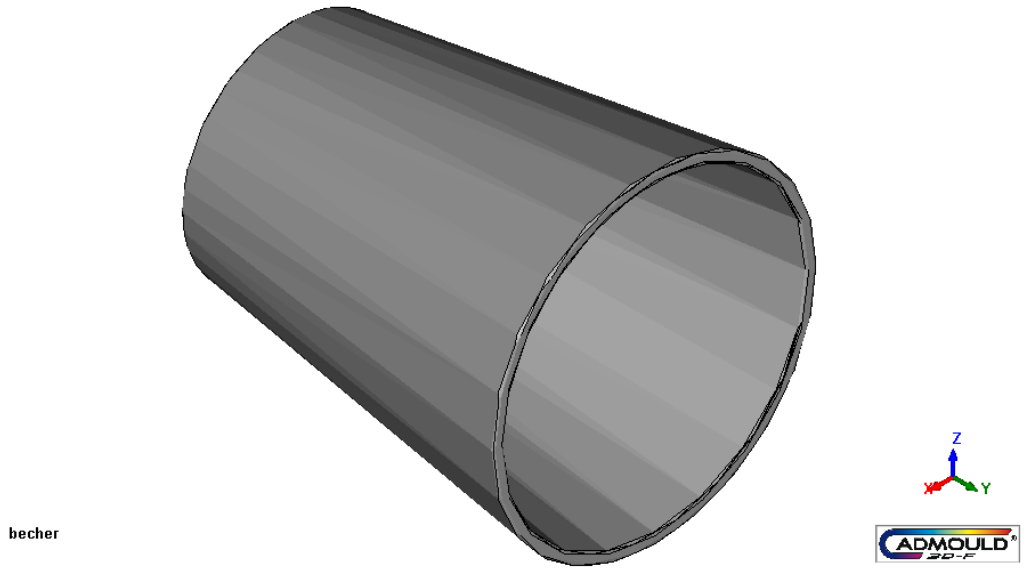
(5) Wait for the geometry to load and appear on the screen.

⇒ The window "Scale the part to millimeters" appears.

(6) Compare the size of the bounding box of your part with the units given on the left side in this window. Click the unit which gives the right size of your part and then "OK", or scale the part to your needs in the scale window "?" and then click “OK”.



(*See* chapter: Scale)



Example: becher.stl

Note: If an STL-file contains more than one part, a message will appear "n parts extracted from "rubbish.stl"!". If the user declines this as an error (in the stl), he can show or hide the objects with the object list on the card "Visibility" in the dialog "View Options" With the dialog "Delete" he can delete erroneous parts easily.

4.1.2 Replace

In case there is already a part loaded it may be necessary to replace it by another part.

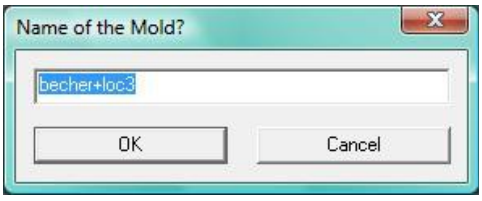
1) Click on "Import STL File".
   ⇒ The window "Open" appears.
   Follow the work steps in **15.1.1.1** until Point 4)
   ⇒ The window "Restart?" appears.

2) Click <Replace> and continue in **15.1.1.1** starting with step 5)

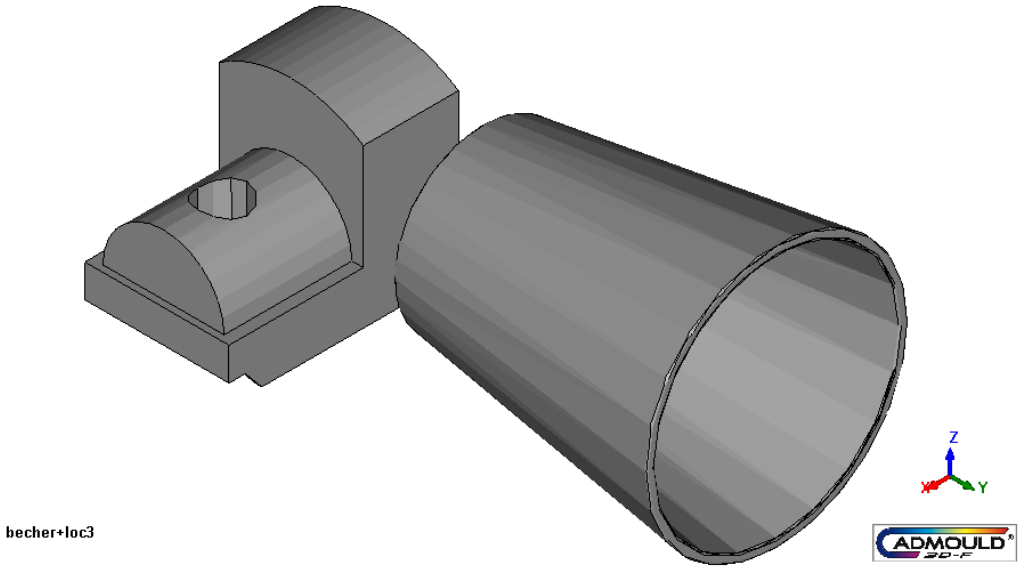
4.1.3 Merge

In case you want to simulate a multi-cavity mold this step is necessary.

1) Click on "Import STL File".
   ⇒ The window "Open" appears.
   (Follow the work steps in **15.1.1.1** until Point 4)
   ⇒ The window "Restart?" appears.
2) Click <Merge> and select a further part to be loaded.
3) Chose a name for the combined "mold".



Example: becher+loc3



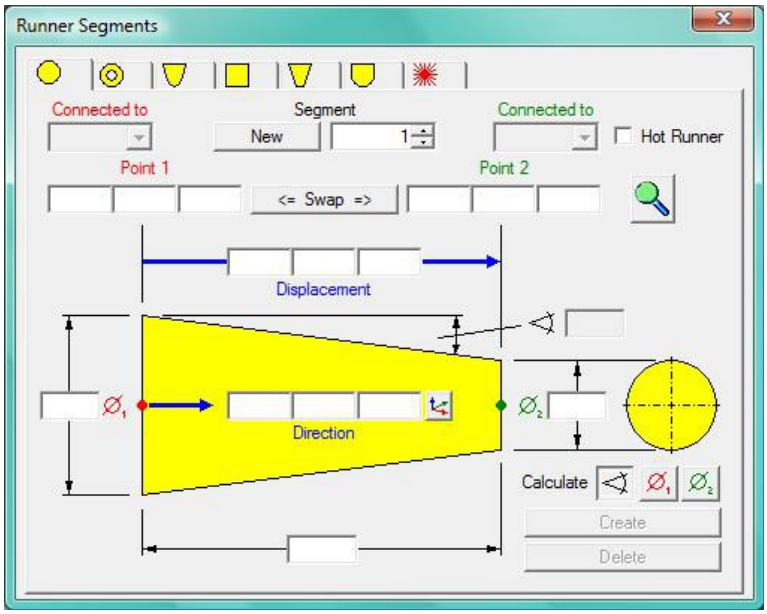
4.2 The runner system

4.2.1 Runner segment

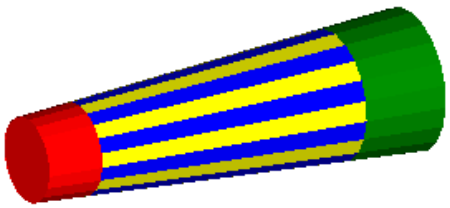
Click "Runner Segment".

⇒ The window "Runner Segments" appears on the screen.

To input a segment several data are necessary. These form the information of the vertices of starting and ending point (color coded red and green) as well as the diameters at each end. There is more than one way to define these data. Use what fits your needs best. Edit fields are filled in automatically as soon as they are redundant to your inputs. E.g. defining point 1 and the displacement to point 2 makes the program fill in point 2, the length and the direction.



Example: Runner segment



4.2.2 Select Function

All coordinates can be defined by a mouse-click.

Position the cursor in the field you want to get the coordinates for. Keep the <Ctrl> key pressed and click on the part or onto that end of an existing segment (red or green) where you want to position the runner segment. The coordinates on the part or of the center line's end of the runner segment are transferred to the input field.

Any displacement or direction can be grabbed in the same manner, tangents and the bending direction for linking bends and hoses (heating/cooling segments) with already existing heating/cooling segments can be transferred to the input field, e.g. the vector normal to a surface or a segment's start to end displacement. Picking the colored end of a segment gives the direction of the front face of the segment. The cursor focus must be in one of the X/Y/Z fields prior to selection.

Diameter and length information can be picked accordingly.

Parts and runner segments can be selected with "<Ctrl>+left mouse button" directly at the screen.

Note: You can also define a direction, by putting the letter of a standard direction, like X/Y/Z into the first field of direction. Put in the letter with opposite sign, to turn the direction 180°.

4.2.3 Zoom

Activate this button to show the segment full screen. This makes it easier to find the segment in a complex runner system.

4.2.4 Calculation

For a conical segment by default the angle is calculated based on given length and both ending diameters. You may also give this angle and one diameter and have the missing diameter added.

Click the symbol of the value you want to have calculated.

4.2.5 Hot/Cold Runner

Before creating a segment you can define it as a "Hot Runner" by setting the tag "√ Hot Runner".

A "Hot Runner" is shown in red and yellow stripes. A "Cold Runner" is shown in blue and yellow stripes.

For every hot runner segment can later in the process parameter window a temperature be assigend.

Please see also: Chapter Heterogeneous Hot Runner Temperature.

4.2.6 Update

1) Click the arrow “New” or “Existing” down or up to open a segment you want to update.
2) Change the parameter(s).
3) Click < Update >.

If you don’t click <Update> the changes will be not effective.

4.2.7 Delete

1) Click the arrow “New” or “Existing” down or up to open a segment you want to delete.
   ⇒ The selected part or segment is highlighted red.
2) Click on <Delete>.

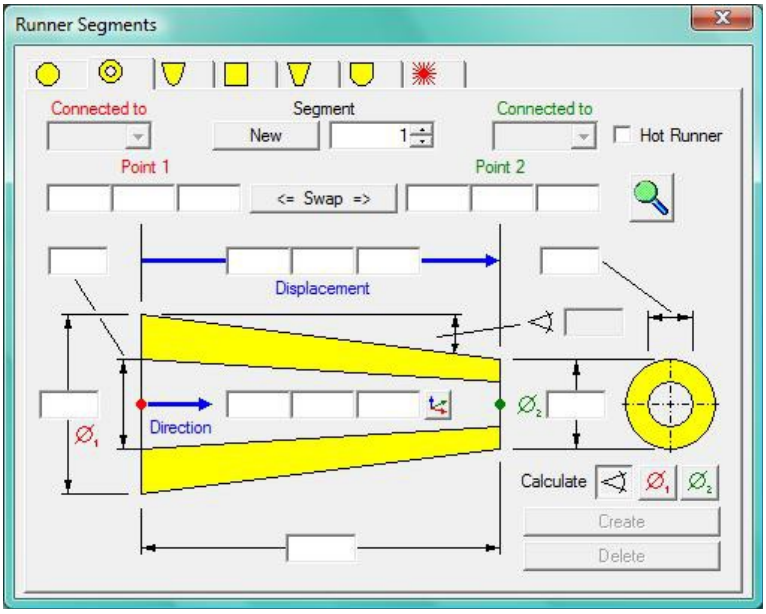
4.2.8 Further cross-sections of a runner segment

All runner segments (square, rounded square, trapezoid, rounded trapezoid, annular shape) you create appear as cylindrical or bevel runner segments on the screen, as the program uses the hydraulic diameter in simulation. Furthermore so far it is not possible to define or display an orientation of non-symmetric shapes.

(See the technical and/or theory manual)

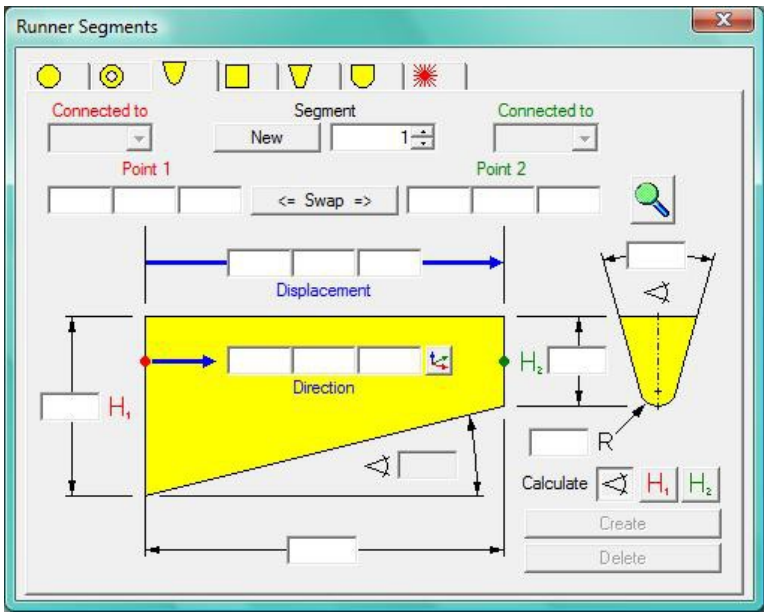
4.2.8.1 Needle valve nozzle/ annular shape

Enter the inside and outside diameter on both sides (red and green), displacement, direction + length and the coordinates of “Point 1” (red) and “Point 2” (green) of the runner segment.



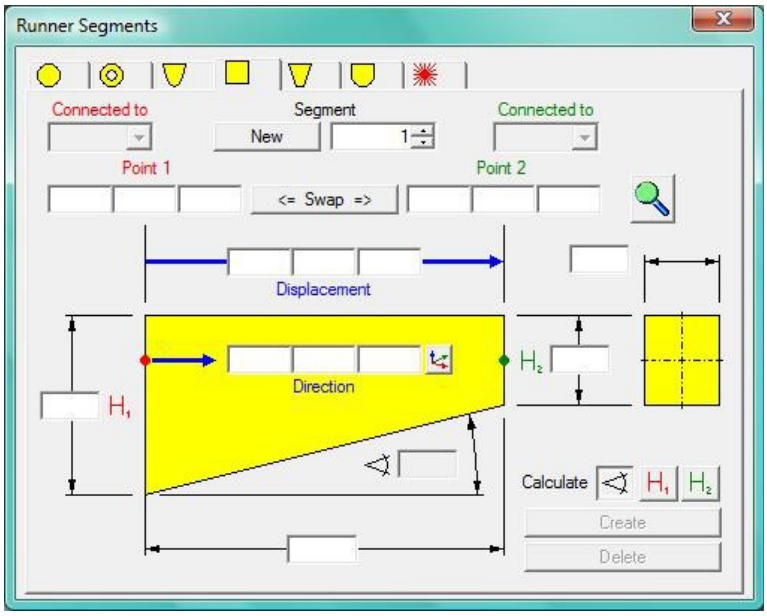
4.2.8.2 Rounded trapezoid

Enter the height on both sides (red and green), angle of retraction, radius, opening angle, displacement, direction and the coordinates of “Point 1” (red) and “Point 2” (green) of the rounded trapezoid.



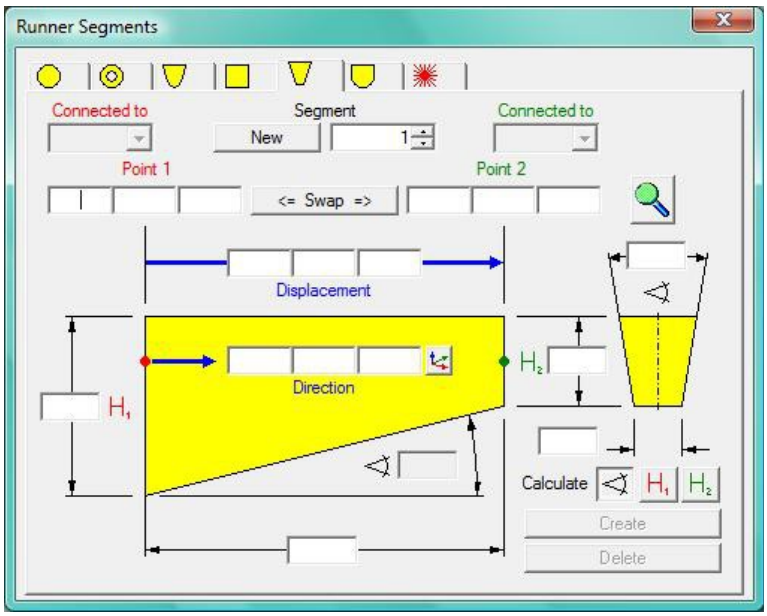
4.2.8.3 Square

Enter the cross-sections on both ends (red and green), angle of retraction, displacement, direction and the coordinates of “Point 1” (red) and “Point 2” (green) of the square runner segment.

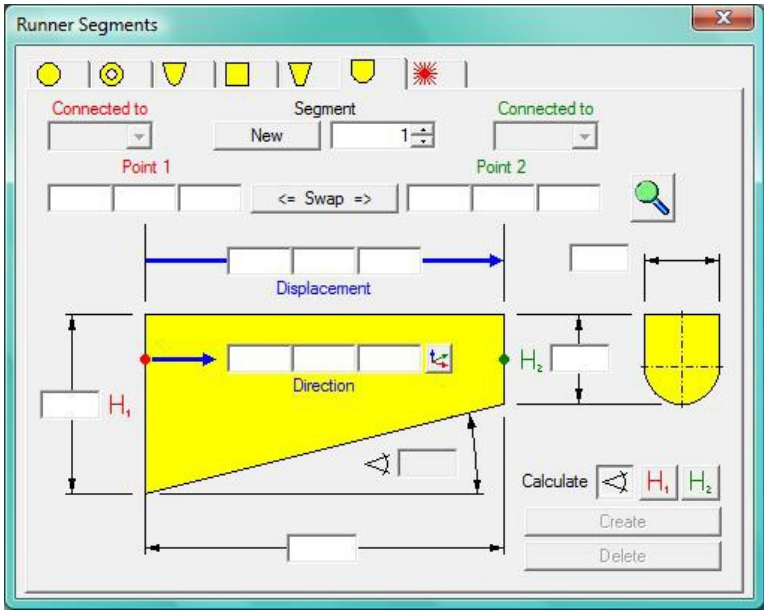


4.2.8.4 Trapezoid

Enter the height on both sides (red and green), angles of retraction and opening, width, displacement, direction and the coordinates of “Point 1” (red) and “Point 2” (green) of the trapezoid runner segment.

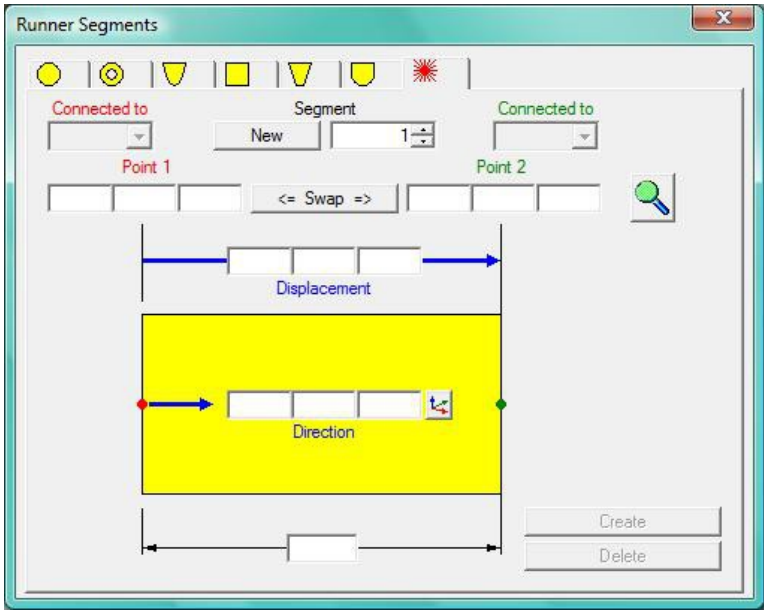


4.2.8.5 Rounded square



Enter the height on both ends (red and green), width, angle of retraction, displacement, direction and the coordinates of “Point 1” (red) and “Point 2” (green) of the runner segment.

4.2.8.6 Virtual Runner Segments



Virtual Runner segments make it possible to connect an existing runner system with a part, without a direct contact at the section. The melt will be transported through the segment without delay and pressure loss.

These virtual segments are applicably e.g. at a test dimensioning, if an approximate shape of the runner is known, but the exact balancing connection is searched. Then different versions can be analysed without much changing time and effort. Finally, the best version should be checked again with a runner, build with regular runner segments.

Note: A virtual runner segment is displayed as a thin, yellow-white striped segment. The displayed diameter serves only for a better recognisability.

4.2.9 Example of a runner system

It is advised to build the runner system off the part to guarantee a correct connection to the part. Furthermore think in segments, i.e. from one change of direction or diameter to the following.

4.2.9.1 First runner segment

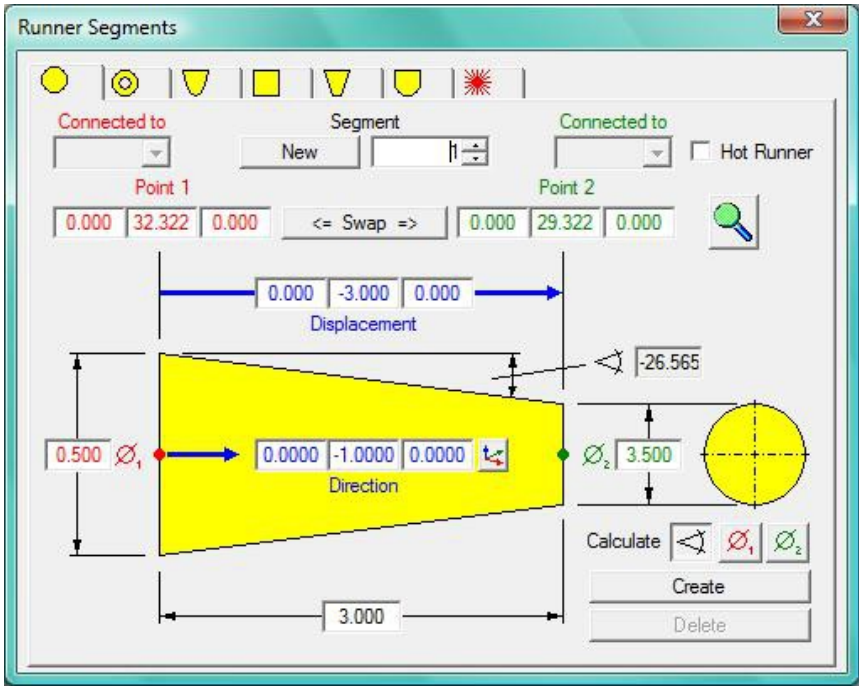
1) Click "Runner segment".

   ⇒ The window "Runner Segments" appears.

1) Set the cursor in one of the three fields of <Point 1> (red).
2) Keep the <Ctrl> key pressed and click on your part where you want to link the runner segment with the part. Point 1 will receive the coordinates of the selected surface point. To get the coordinates of the second point, you can either:

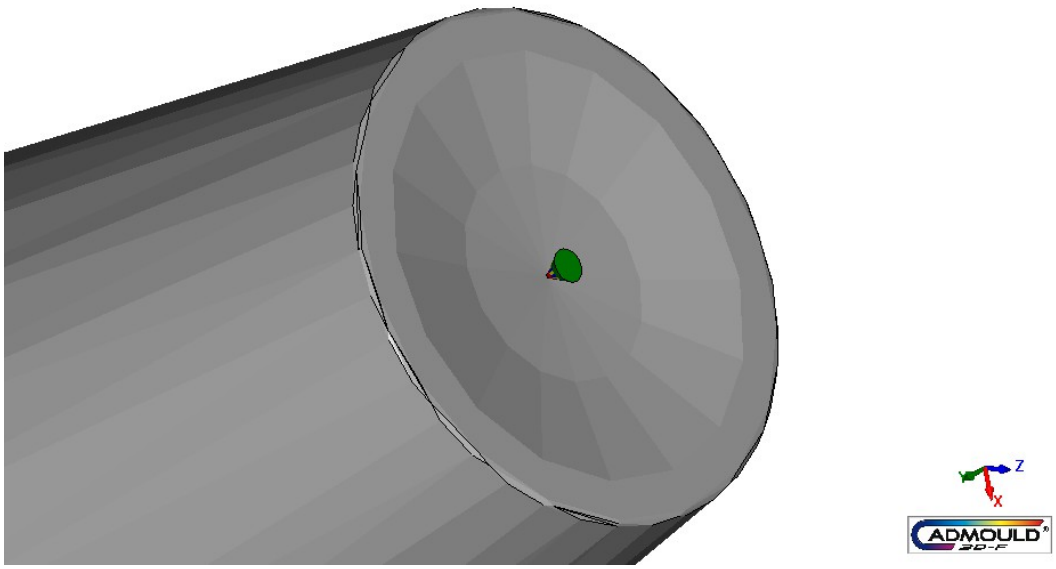
- ♦ enter them directly,
- ♦ enter the direction (e.g. by mouse click) and the length,
- ♦ enter the displacement.

3) Enter the parameters of the runner segment.
4) Before you click <Create> leave the last field by pressing the <Tab> key. (If the option <Create> still doesn't appear, click the <Tab> key more times. Make sure to define the runner segment completely.)



5) Click <Create> to get the segment on the screen.

As you can define both diameters of a segment the sketch in the window "Runner Segments" is not representative by its dimensions. The diameter at <Point 1> (red) may be smaller than that at <Point 2> (green) opposing the sketch.



The left diameter (red) has been projected onto the cups bottom.

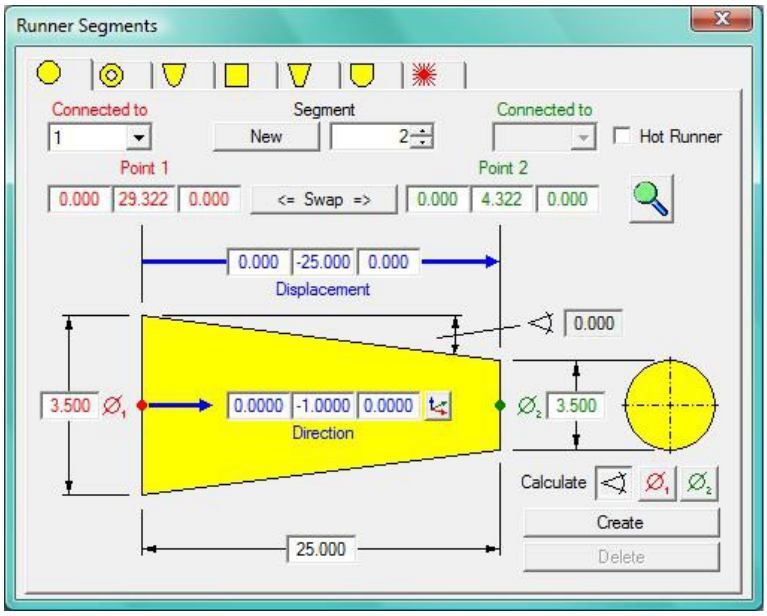
4.2.9.2 Second runner segment

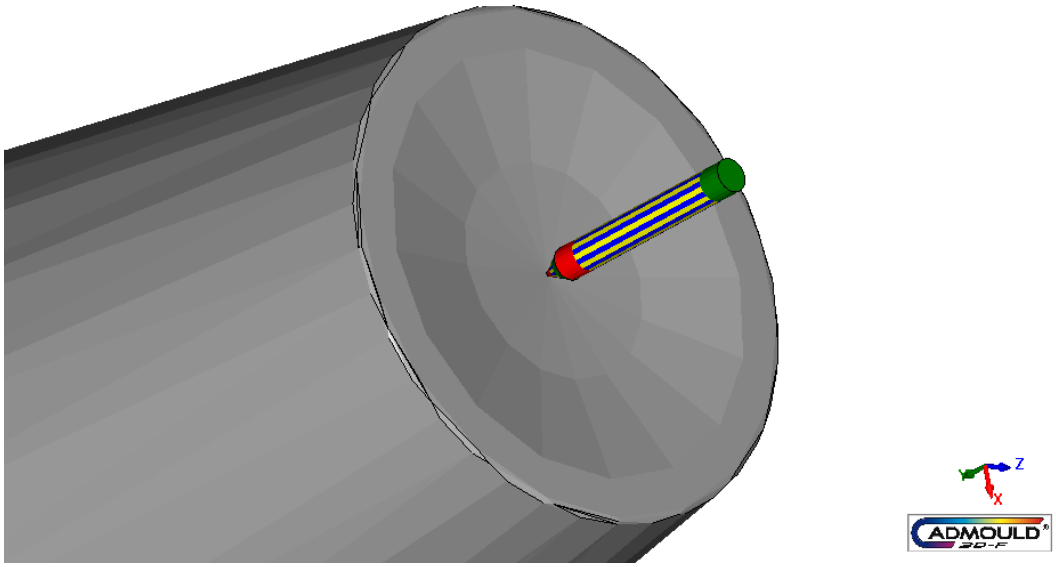
A new runner segment can only be defined if the segment number input field at the center top reads “new”.

Already existing segments (*the input field reads* “Existing” *and a segment is highlighted red*) can be updated or deleted.

Define the coordinates by mouse-click, or you proceed as follows:

1) Click the arrow “New” down to get the existing, last created segment.
2) Swap the coordinate of the <Point 2> (green) of the previous segment.
3) Enter the parameters of the second segment and leave the field by clicking <Tab> key. If you didn’t define a new segment, i.e. didn’t enter displacement or direction, you will receive an error message, as you are up to creating a identical segment.
4) Click the arrow “Existing” up to create a new segment.
5) Click on <Create>.

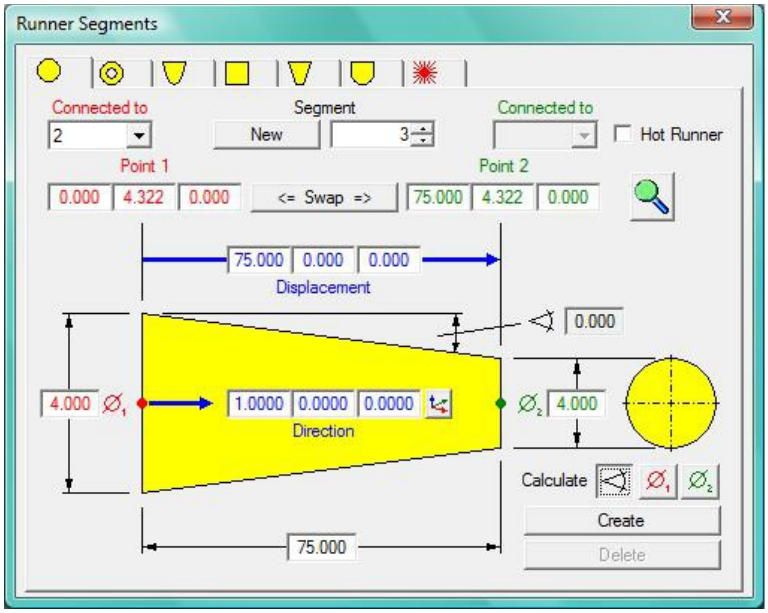


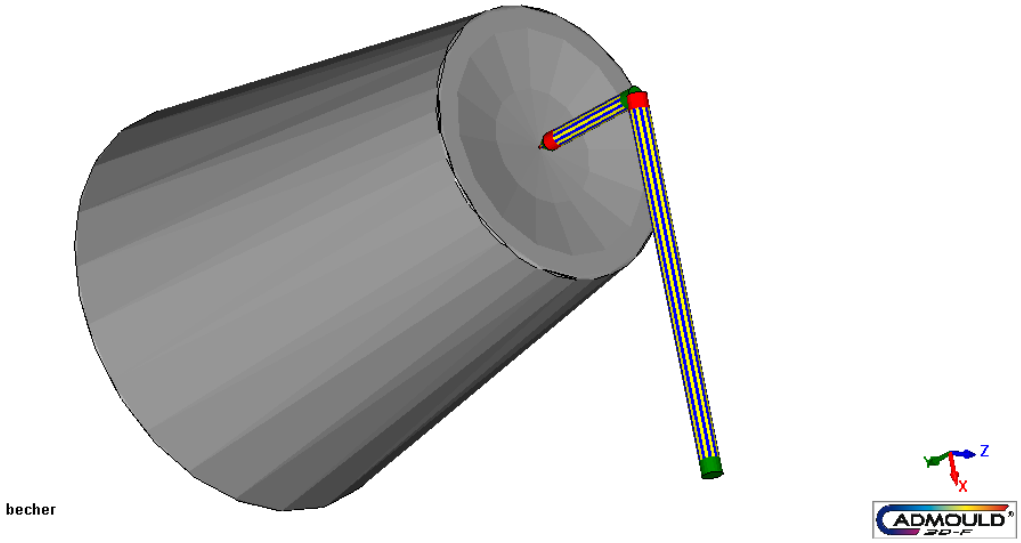


4.2.9.3 Third runner segment

Define the coordinates by a mouse-click (*See* chapter: "Runner segment"),
or proceed as follows:

1) Click the arrow "New" down to get the existing, last created segment.
2) Swap the coordinate of the < Point 2 > (green) of the second segment.
3) Enter the parameters of the third segment and leave the field by clicking <Tab> key.
4) Click the arrow "Existing" up to create a new segment.
5) Click on <Create>.





4.2.10 Construction assistance

It´s now possible to move runner segments by drag and drop in space. To do this, select the segment with CTRL + middle mouse button. If you select the segment on the boundary surface, you will move the whole segment. The direction in space and the length won´t change.

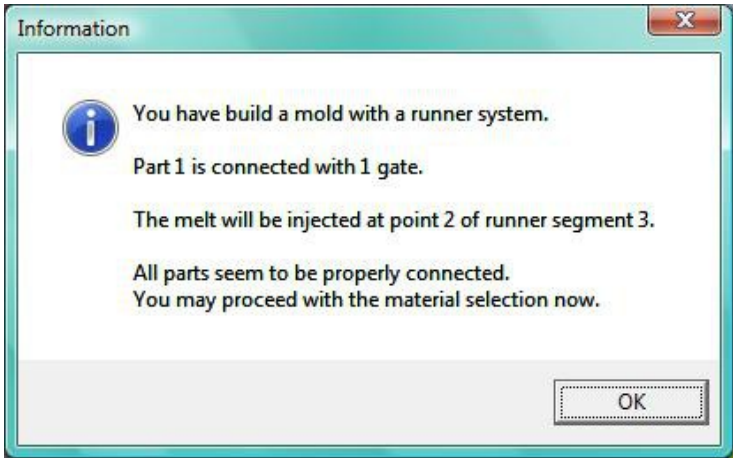
If you select the end of a segment, only this end will move. Take care, direction and length of the segment can change now. If two ends of segments are visibly together, they will be fixed together, if you loose the button.

After the displacement, you have to click "Actualize" to accept the changes.

4.3 Gate

Click the icon “Gate” to get information about the connectivity between the runner segments and the injection point.

Click “OK”. If all parts are properly connected, continue. Otherwise correct the runner system.

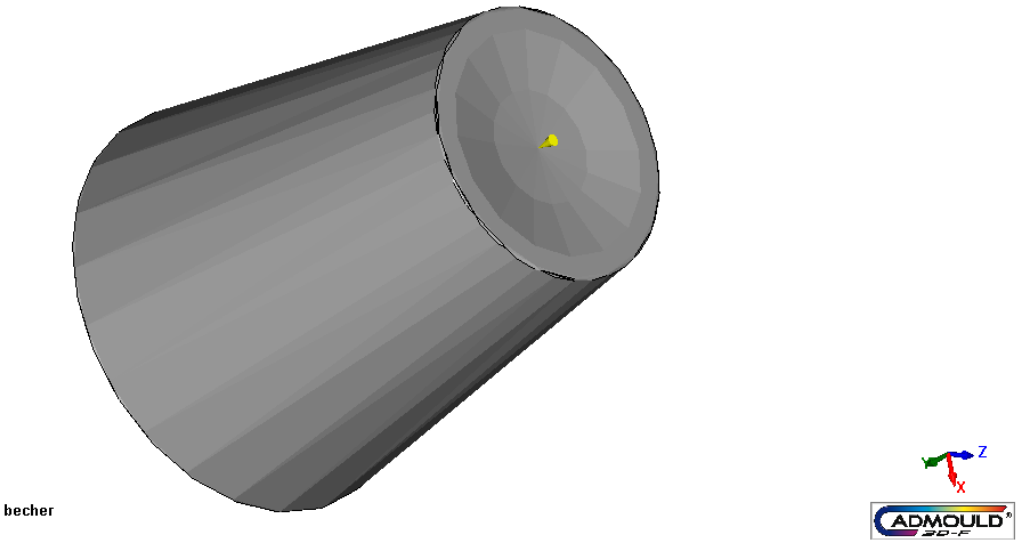

Information
You have build a mold with a runner system.
Part 1 is connected with 1 gate.
The melt will be injected at point 2 of runner segment 3.
All parts seem to be properly connected.
You may proceed with the material selection now.
OK

Note: Open runner ends are colored red resp. green, even if no dialogue is open where runner end points can be digitized.

4.3.1 Define a gate location

If you have not built a runner system the following window will appear on the screen and you can define one or more gate locations without a runner system.

Gates appear on the screen as a yellow cone on the part.



Example: Cup with one gate

Note: If you want to define more than one gate location, please consider chapter Virtual Runner

4.3.1.1 Enter the Gates

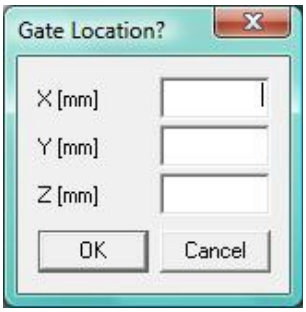
a) by a mouse-click

b) by keyboard

   a) by a mouse-click

Keep the <Ctrl> key pressed and click the location on the part where you want to gate it. The coordinates of the gate will be indicated in the window "Gate Locations".

b) by keyboard



(1) Click "Add".

(2) Wait, until the window "Edit Gate Locations" appears on the screen.

(3) Indicate the coordinates as x, y, z components.

⇒ If the indicated point does not fit to the part surface, the desired gate location will be projected onto the part surface in view direction.

(4) Click "OK".

Note: If you know two of three coordinates, you can leave the third coordinate empty. CADMOULD® 3D-F will find it by itself.

You just have to put in a + or –, to define the direction from which CADMOULD® 3D-F has to drop a perpendicular onto the part surface. Eg: (14.5, 0, +) The input '+' will let the program drop a perpendicular from the point (14.5, 0, Zmax) in negative Z-direction from above down onto the part. If you want to define the gate on the bottom, you put (14.5, 0, -) in, without rotating the part (which was necessary in earlier versions, because the projection of points in viewing direction could find only visible points). Then the program will drop a perpendicular from the point (14.5, 0, Zmin) in positive Z-direction from below up onto the part.

4.3.1.2 Edit

(1) Mark the coordinates by clicking them.

(2) Click "Edit".

(3) Wait until the window "Gate Location" appears on the screen.

(4) Enter modified coordinates.

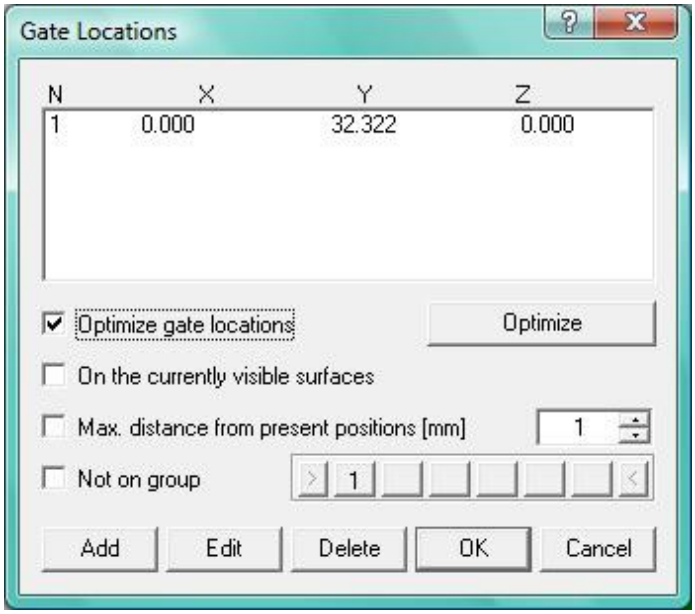
(5) Click "OK".

4.3.1.3 Delete

(1) Mark the coordinates by clicking them.

(2) Click "Delete".

4.3.1.4 Optimizing the gate locations



CADMOULD® 3D-F provides to you the possibility, to optimize the gate locations chosen by you.

Therefore, you have to activate the option <Optimize gate locations> in the window <Gate locations>. The option <Permit crossing of the part> means, that CADMOULD® 3D-F involves the opposite side of the part also. This could be helpful, if you want to gate at a root of fin and does not matter, on which side the gate is set.

The next step is choosing your options and then click <Optimize> to start the gate optimization.

If you agree to the new locations click <OK>, otherwise click <Undo> to restore the old locations.

Note: The optimization can only be run, if the mesh of the part is already prepared.

4.3.1.5 Display Flow Length / Thickness

The function <Display Flow Length / Thickness> helps you to define the number of gates and their positions. To use this function, you need a prepared mesh.

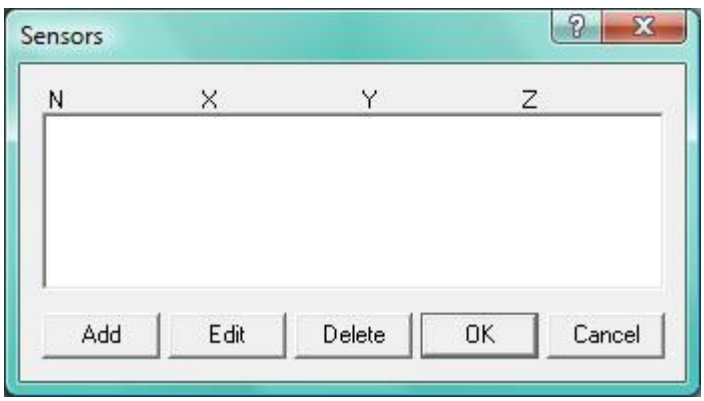
4.4 Sensors

(Optional)

The instruction “Sensors” offers the possibility of definition points on your part you want to proof exactly. After execution of the simulation the defined “Sensors” appear as a result group in the window “Result Selection”.

1) Click on the button “Sensor Editor”

   ⇒ The window: “Sensors” appears on the screen.



Keep the <Ctrl> key pressed and click on the part where you want to position the “Sensor”. The coordinates of the point are indicated in the window. You may also enter the coordinates by keyboard.

- Click <Add> to enter the coordinates by keyboard.
- Click <OK> to accept the “Sensor”.

♦ You can also edit or delete the coordinates of the “Sensor”.

You also find the command "Sensor Editor" in the pull down menu "Simulation" under the instruction “Sensors”.

An alternative possibility instead of setting “Sensors” is to use the post processor function for measuring results.

(Consider please the chapter “Local Value vs. Time”)

4.5 Cascade Injection Molding

(optional component to CADMOULD® 3D-F FILL)

Weld lines can lower the mechanical characteristics of your part substantially or are a cosmetic defect. Cascade injection molding allows to manufacture extended parts without visible weld lines, and/or to shift the weld lines purposely into non restricted part areas.

The melt feed is made by a hot runner system, the filling is controlled by opening or shutting off individual or several valve gate nozzles.

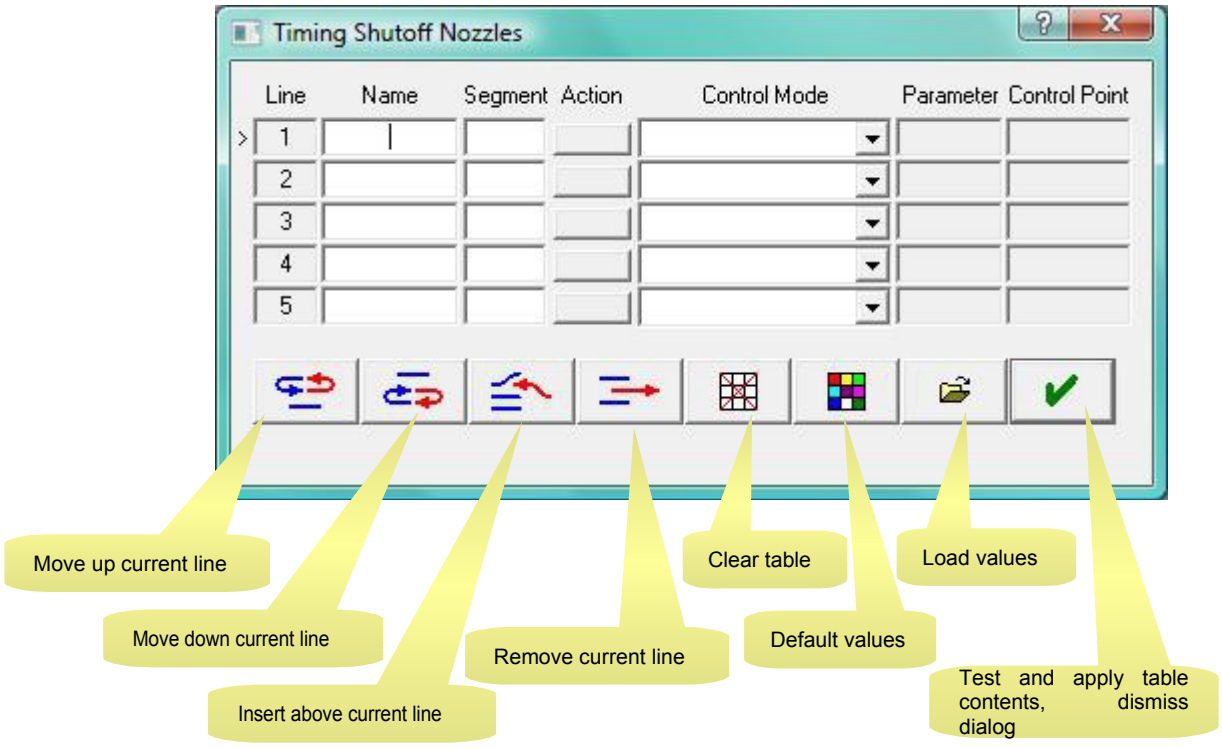
The packing pressure is applied by re-opening of the nozzles.

(1) Design a hot runner system.

(2) Examine your hot runner system, by clicking on „Gate Editor“. Correct the runner system in case of errors.

(3) Click on "Shutoff Nozzles".

⇒ The window "Timing Shutoff Nozzles" appears on the screen.



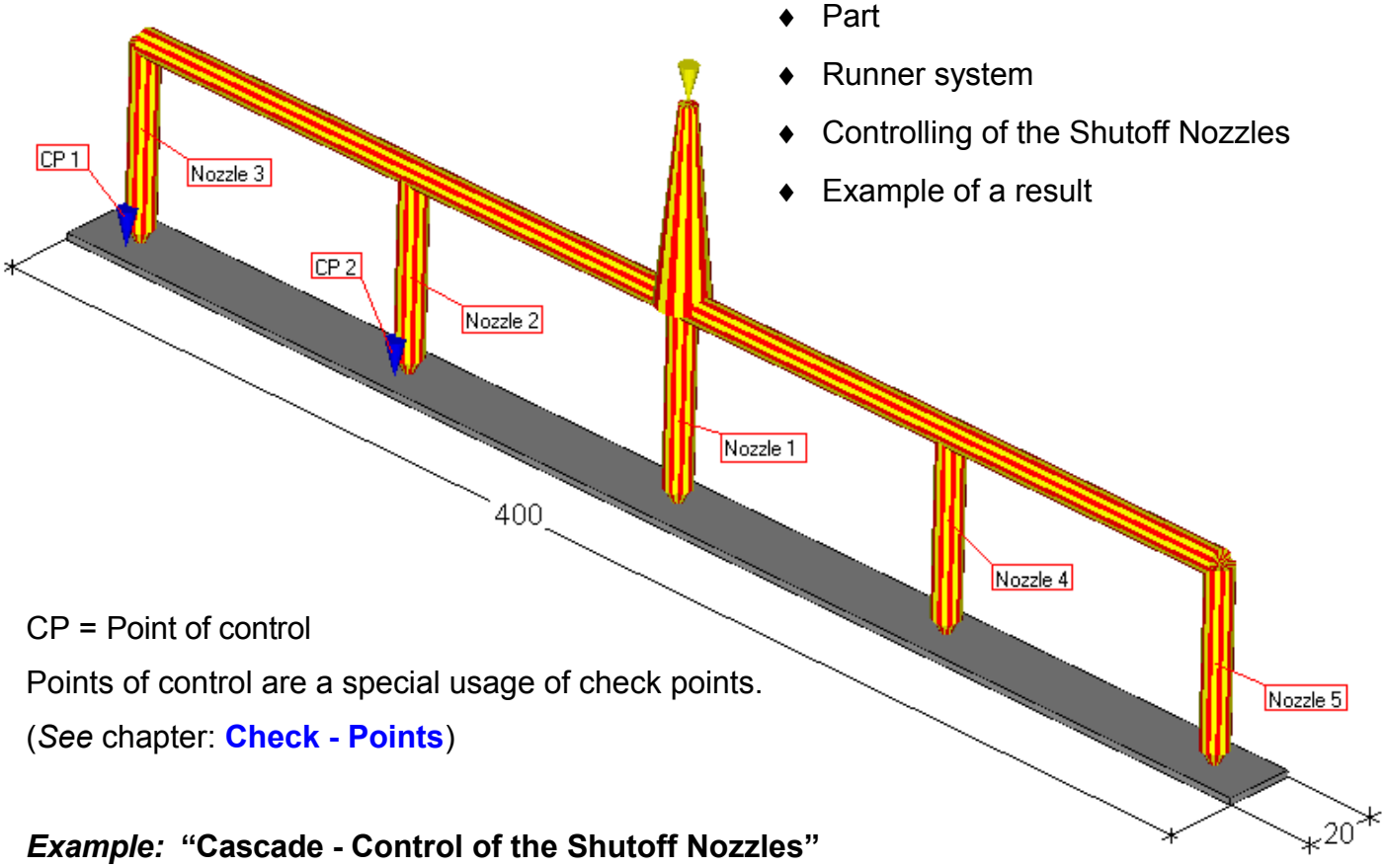
(4) Click on <**Default values**>.

(5) Enter the control of the Shutoff Nozzles.

(*See* example: **"Cascade - Control of the Shutoff Nozzles"**)

(6) Exit the input by clicking on <Test and apply table contents, dismiss dialog>.

Example: **"Cascade" part, runner system**



- Part
- Runner system
- Controlling of the Shutoff Nozzles
- Example of a result

CP = Point of control

Points of control are a special usage of check points.

(*See* chapter: **Check - Points**)

Example: **"Cascade - Control of the Shutoff Nozzles"**

Column: Description:

Line: Line number

Designation: The nozzle designation can be selected freely.

Segment: The segments, which are to function as shutoff nozzles, can be selected with "**Ctrl + Left Mouse Button**" on the runner system, or be registered automatically by clicking on <**Default values**>. The segments are numbered automatically. They are marked red.

Action: Single click on the button determine whether the shutoff nozzle should open or should close.

Control type: The control type of the shutoff nozzle can be selected in the list field (column: Control type) by mouse-click.

Point of control: Points of control can be selected with "**Ctrl + left mouse button**" on the part. Selected points of control are marked red.

„Active line": The marking " **>** " on the left of the column "line" marks the active line.

„Cascade"– example of a result shows the function of the following control.

= Nozzle open

= Nozzle close

Filling Time = 1 s

Packing Time = 5 s

Line	Designation	Segment	Action	Control type	Parameters	Point of control
1	Nozzle 1	10		1 Time [s]	0.0	
2	Nozzle 1	10		3 CP reached		2
3	Nozzle 2	11		3 CP reached		2
4	Nozzle 4	14		3 CP reached		2
5	Nozzle 2	11		3 CP reached		1
6	Nozzle 4	14		3 CP reached		1
7	Nozzle 3	12		3 CP reached		1
8	Nozzle 5	15		3 CP reached		1
9	Nozzle 1	10		2 Level [%]	99.0	
10	Nozzle 2	11		2 Level [%]	99.0	
11	Nozzle 4	14		2 Level [%]	99.0	

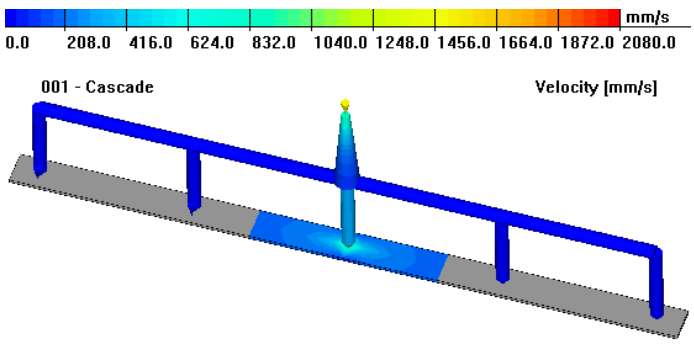
Notice: The controlling of shutoff nozzles should always contain a line, which opens a shutoff nozzle on the time = 0,0 s (see line 1) or with the level = 0,0 %, otherwise directly with the beginning of simulation the flow front freezes.

Example: **"Cascade - example of a result"**

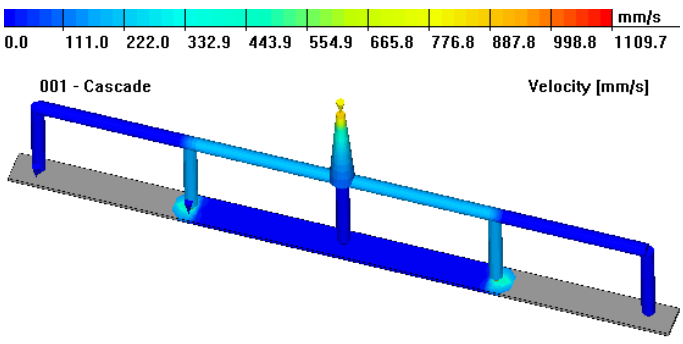
Process parameter:

Filling Time	= 1 s	Ejection Temperature	= 120 °C
Melt Temperature	= 250 °C	Packing Pressure	= 75 %
Hot Runner – Temperature	= 250 °C	Packing Time	= 5 s
Wall Temperature	= 40 °C		

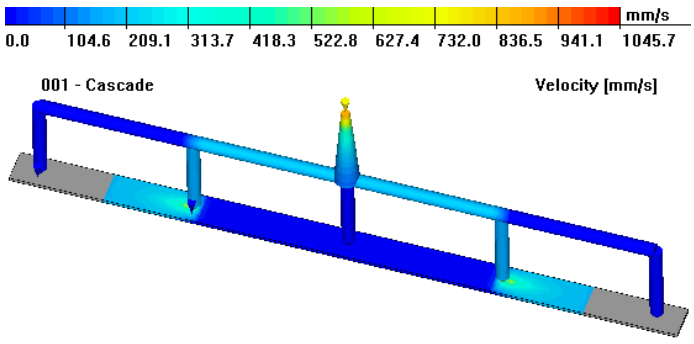
Pattern 1: Level 30%



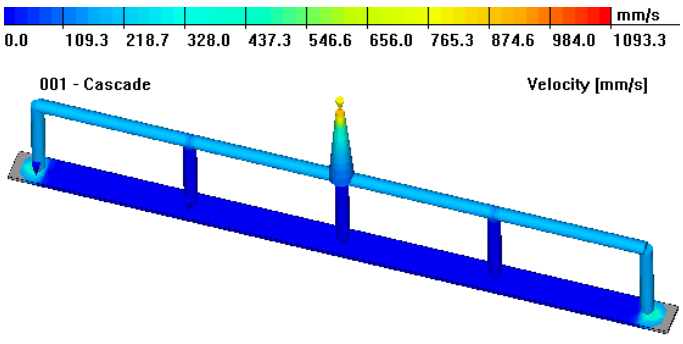
Pattern 2: Level 51%

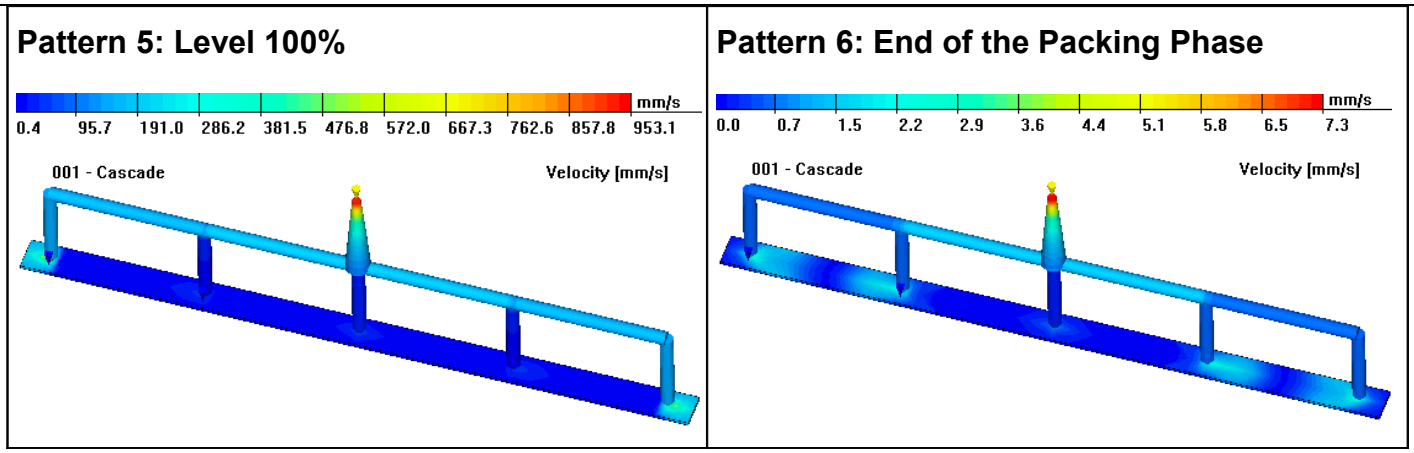


Pattern 3: Level 75%



Pattern 4: Level 97%

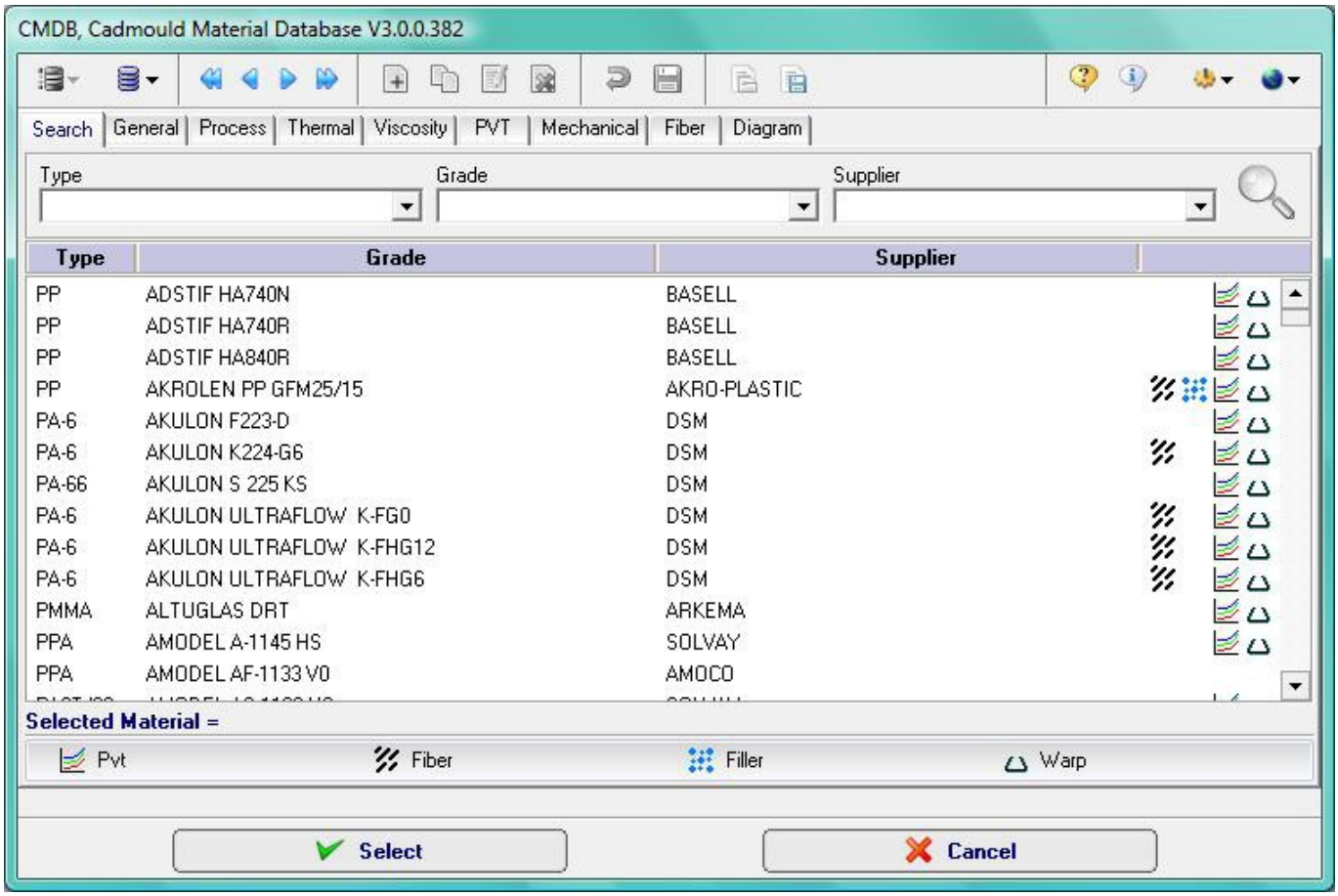



Pattern 5: Level 100%
mm/s
0.4 95.7 191.0 286.2 381.5 476.8 572.0 667.3 762.6 857.8 953.1
001 - Cascade
Velocity [mm/s]
Pattern 6: End of the Packing Phase
mm/s
0.0 0.7 1.5 2.2 2.9 3.6 4.4 5.1 5.8 6.5 7.3
001 - Cascade
Velocity [mm/s]

4.6 Select a material

(1) Click "Material Selection".

⇒ The window "CADMOULD® 3D-F material data base" appears on the screen.



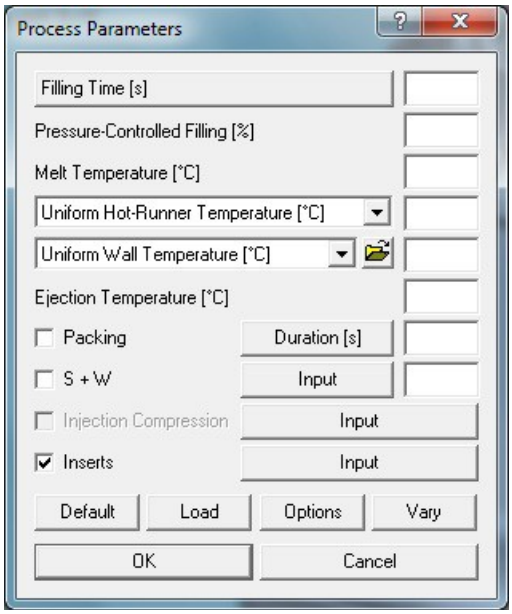
(2) Find the material you want to use.

(3) Mark the desired material by clicking.

(4) Click "Select", if you marked the correct material.

Double-clicking in step (3) has the same effect.

4.7 Define the process conditions



(1) Click "Process Parameters".

⇒ The window "Process Parameters" appears on the screen.

(2) Define the Filling Time, the Switch-to-Packing Level, the Melt Temperature, the Hot-Runner Temperature, the Wall Temperature and the Ejection Temperature,

or

click "Default", if you like to use proposed values for the selected material,

or

click “Load”, if you like to see a selection of RM1/DES/SYM-files which can be used for the actual project.

If it is possible, activate (√) the simulation of the packing phase and shrinkage and warpage prediction.

(3) Change the values, if you do not want to accept them.

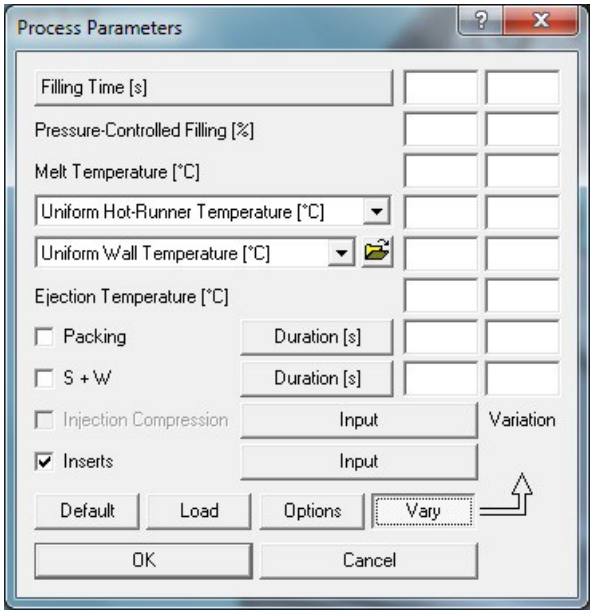
(4) Click "OK", if you accept the modified parameters or continue to profile the injection in the next section.

4.7.1 Vary the process conditions

You have the possibility to let CADMOULD® 3D-F execute a Design of Experiment, where CADMOULD® 3D-F computes with different process conditions, which were defined by you before. To use this function follow the next steps:

1) Click <Vary> in the window "Process-Parameters".

⇒ The window "Process Parameters" is extended by the column "Variation".



2) Enter always the positive variation of the median in the new column.

e.g.: 0.2s with a median of 1s results a variation between 0.8s, 1s and 1.2s of Filling Time.

3) When you click on "Go" (Simulation Start) after entering the "Simulation Description", a new folder is created, where all simulations of this DOE will be saved.

4) All simulations will automatically be written in the task list, which will then be opened. Then you only have to click "Go" in the task manager to start the calculation in batch mode.

The simulation will only start automatically, if the module “Batch” is licensed. Otherwise you will have to start the simulations from the created folder manually.

(Please see chapter: "Stack and compute projects")

Note: The combination of N variables with 3 possible conditions, would cause 3^N simulations, if proceeded naive. When using a method, developed by Mr. Taguchi, the number of necessary simulations is reduced.

N	S	T
1	3	3
2	9	5
3	27	5
4	91	9
5	243	9
6	729	9
7	2187	9

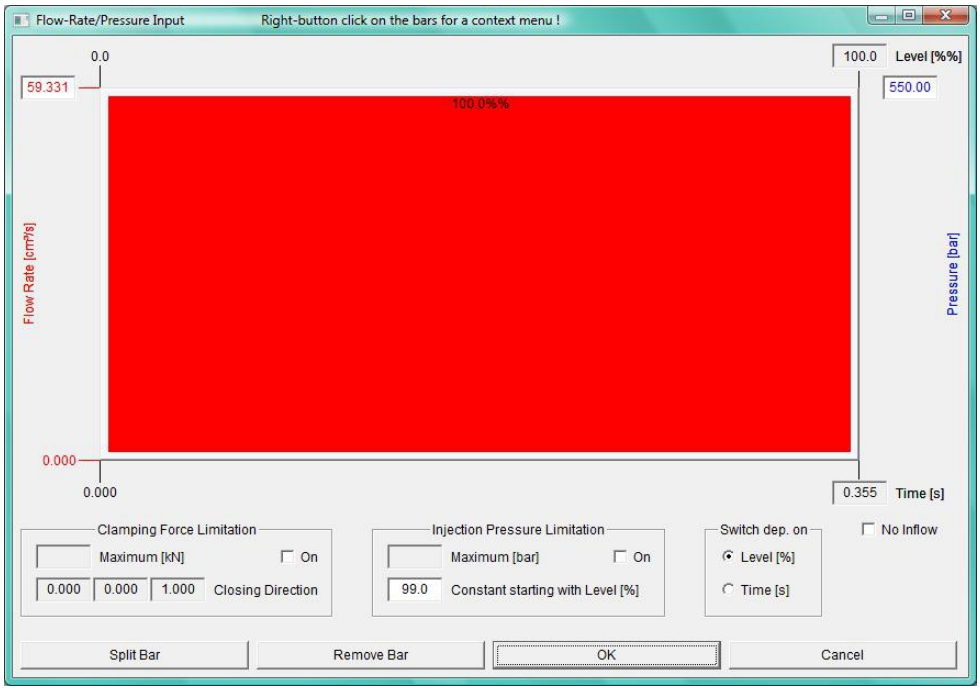
So, if all 7 possible process parameters are varied, only 9 simulations are necessary instead of 3^7=2187 simulations of the naive way.

Note: You find the results of the variation in the menu <Results> under <Variation>.
(Please see chapter "Variation Results")

4.8 Flow Rate / Pressure Input Profile

1) Click “Process Parameters”.
   ⇒ The window “Process Parameters” appears on the screen.
2) Place the cursor on the button: “Filling Time [s]”.
   ⇒ The button changes to “Flow Rate / Pressure Input”.
3) Click this button.
   ⇒ The window “Flow Rate / Pressure Input” appears on the screen.

In the window “Flow Rate / Pressure Input” you see a diagram describing your injection process.



4.8.1 Split Bar

The function: “Split Bar” serves to split an injection process into single injection steps. You can set the switching points precisely and define the process parameter for each step.

You can define a bar by entering a switching point.

1) Click “Split Bar” to active this function.
2) Click somewhere onto the bar (red area) to define the injection step to be split.
3) Drag the switching point with the left mouse-button.

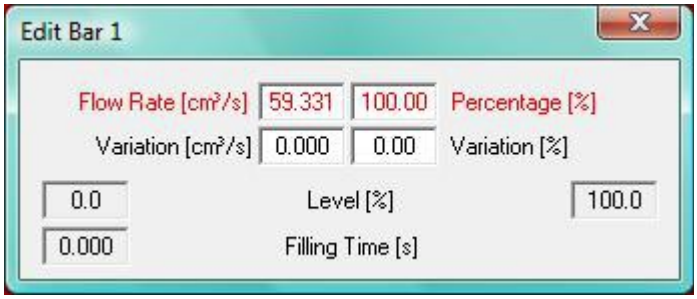
You can also active “Split Bar” by clicking the right mouse-button on the target bar. Select the command “Split” by clicking the left mouse-button.

4.8.2 Edit Bar

You can edit the bars.

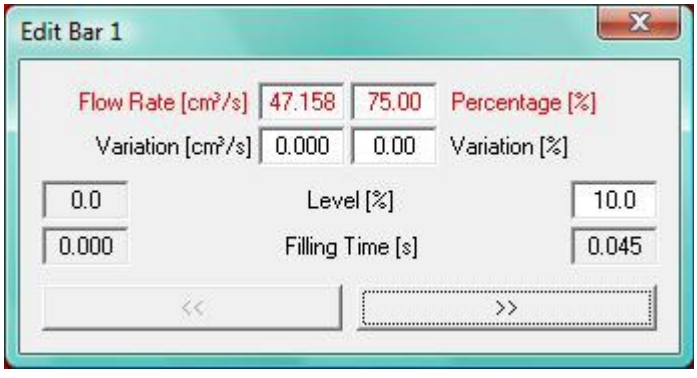
1) Click the right mouse-button on the bar area.
   ⇒ A pull down menu appears on the screen.
2) Click “Edit”
   ⇒ The window “Edit Bar” appears on the screen.

Now, you can edit the flow rate [cm^3/s] and the percentage [%] of the maximum flow rate, resp. also filling level [%] and filling time [s] of the switching to the following step can be defined. The latter inputs must be regarded as alternatives. See below for details.

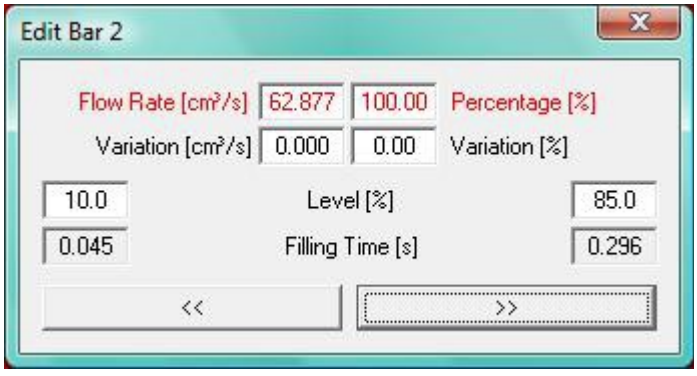


3) Click the arrow right or left in order to select the next bar, if already more than one bar is availabe.

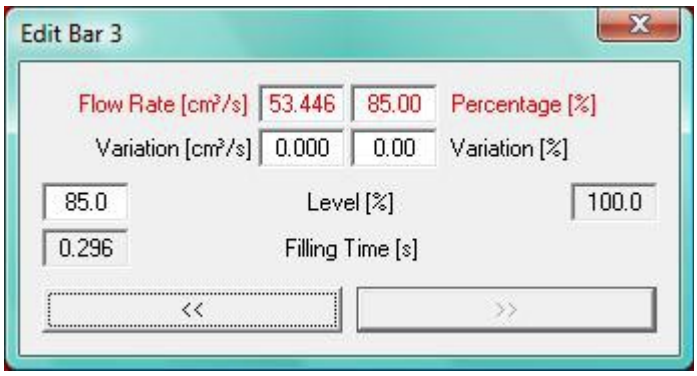
Example: Bar 1:



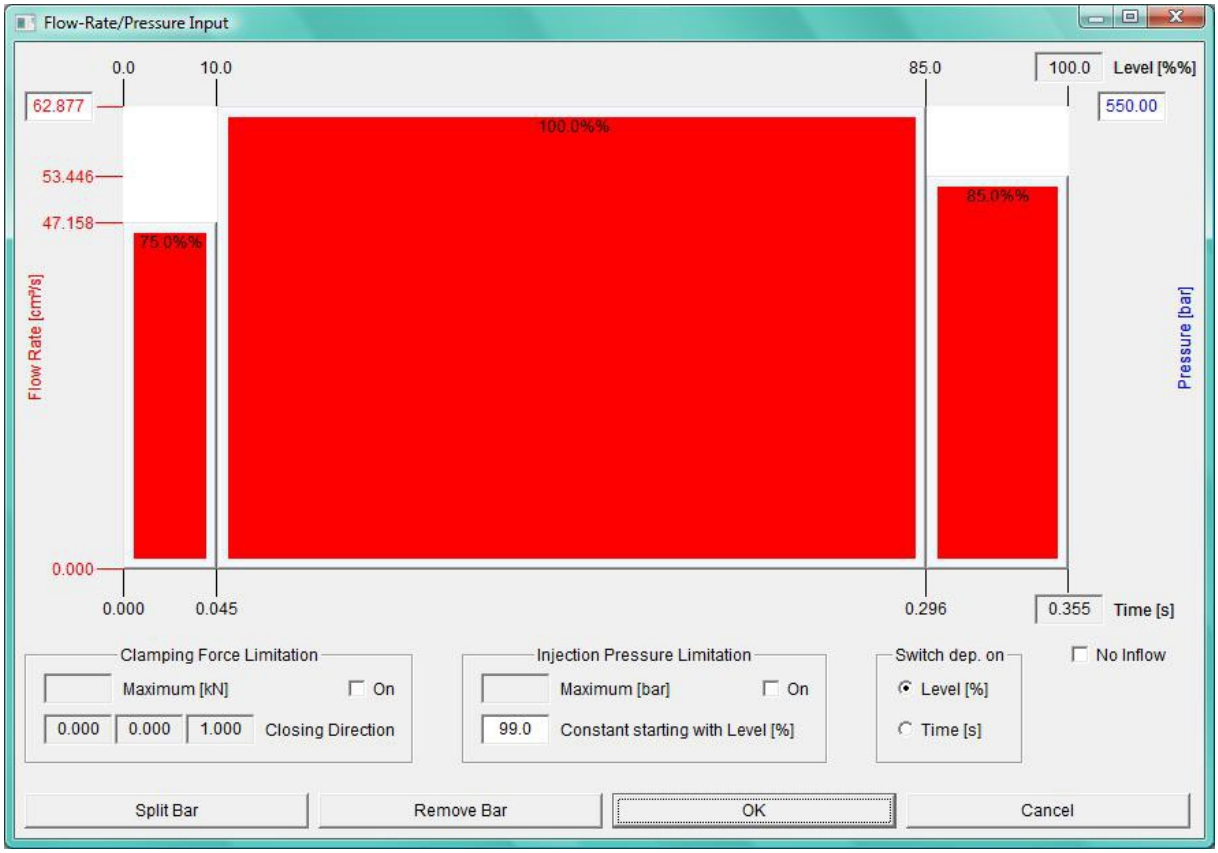
Bar 2:



Bar 3:



The defined flow rate profile:



4.8.2.1 Flow-Rate Variation

CADMOULD® 3D-F gives you the possibility to calculate variants with different flow-rate profiles. Therefore you can define an absolute or percentage variation of the flow-rate in the window of each bar. You enter always the positive variation of the median.

Please see also the chapter: Vary the process conditions

4.8.3 Remove Bar

Use the function “Remove Bar” to delete a bar.

1) Click “Remove Bar” to active this function.
2) Click on the bar you want to delete.

You can also active “Remove Bar” by clicking the right mouse-button in the bar you want to remove and select the command “Remove”.

4.8.4 Clamping Force Limitation

Mark the field: “Clamping Force Limitation” to enter the parameters (√ = On).

Enter the Maximum Clamping Force [kN] and the closing direction.

With this function you can try to run the last phase of the filling process without exceeding the predetermined clamping force. The flow rate, respectively the pressure will be reduced. This reslut can be used for a machine equitably fluting.

But please keep in mind a full supply safety while dimensioning the machine. Also a suggestive packing pressure should be considered.

4.8.5 Injection Pressure Limitation

a) Mark the field: “Injection Pressure Limitation” to enter the parameters (√ = On). Enter the maximum injection pressure [bar]. This pressure will not be exceeded no matter how the profile was defined. For this option to trigger it does not matter at which moment of filling the pressure limit is reached. In case the process runs below the pressure limit this option is not activated.
b) Define the filling level [%] at which the injection pressure is kept constant. This avoids pressure peeks at the end of the filling process similar to switching to packing pressure. The benefit of this option compared to a set pressure is that you need not know the exact pressure.

4.8.6 Switch dep. on

Switch depending on the Time [s] or the Level [%] of filling by marking it (●).

The other scale will no longer be linear. Values given there are for information purpose only. The same applies to the pop-up windows for editing the bars. During a simulation run using compressibility material data (pvT) the non-active switch values may shift due to thermal shrinkage effects.

4.8.7 No inflow

With this option you can simulate a short shot. The filling stops at the entered filling stage.

Remark: You cannot simulate packing when this option is set, because the part must be full.

4.8.8 Pressure

Each step of the filling phase can be switched from a flow rate constant to pressure constant step.

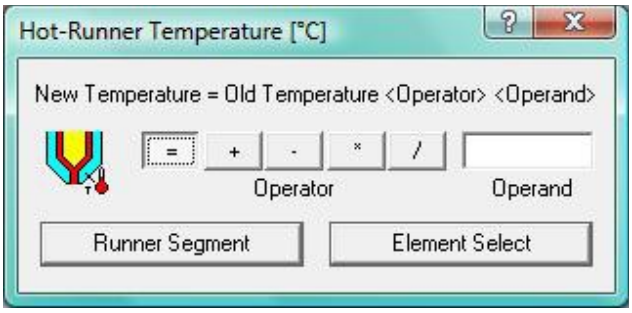
Right-click the bar of the step you want to change. Chose “Pressure” in the menu. The color of the bar changes from red to blue. The scale relevant for pressures is at the right hand side of the diagram. Switching back to a flow rate control is analogue.

4.9 Heterogeneous Hot-Runner Temperatures

(1) From the pull down menu “Uniform Hot-Runner Temperature” choose the option „Hot-Runner Temperature“. Push the button “Input”.

⇒ The window “Hot-Runner Temperature” appears on the screen and the temperature distribution on the hot runner segments is visible on the screen.

(2) You can now define a specific temperature for every hot runner segment. For this you enter an operator and an operand and select the segments resp. the elements like described below:



- Runner segment:

Push the button “Runner Segment” to select segments which have been designed in CADMOULD® 3D-F. Then you select the segments with Ctrl + left mouse button. You close the selection with the right mouse button. Then you only have to apply the changes.

- Element select:

There is also the possibility to select triangle elements, which belong to a prepared runner system in cfe-format. The elements have to be on negative groups to be recognized by CADMOULD® 3D-F as runner system. The element selection is made using the selection tools available on the status bar. The temperature assignment also has to be applied.

4.10 Wall Temperature Distribution

In Cadmould 3D-F you have different possibilities to define a wall temperature distribution.
This is amongst others depending on the modules you have licensed.

4.10.1 Uniform Wall Temperature

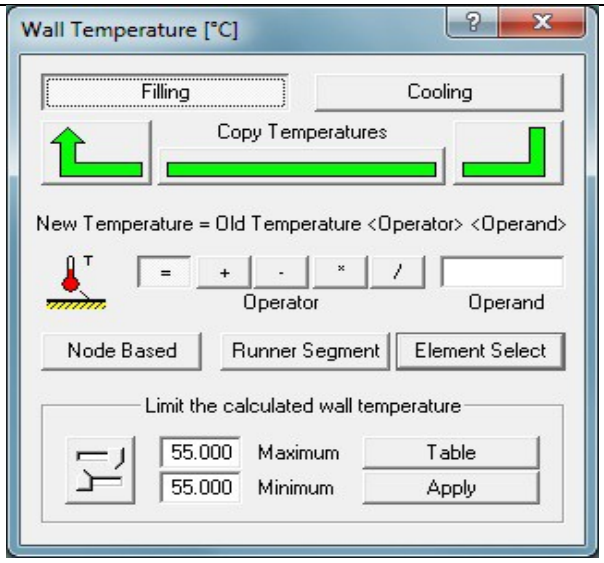
If you don't want to execute a wall temperature analysis or you haven't licensed the module Cool, you can define a uniform wall temperature for the whole cavity.
This temperature is as well for the filling as for the cooling phase.

4.10.2 Assigned Wall Temperatures

You can assign wall temperatures manually.

(1) Click the button "Wall Temperature" from the button bar.

⇒ The window "Wall Temperature" appears on the screen.



(2) You can assign a temperature to every element for the filling and cooling phase. Push the button for the phase you want to assign a temperature.

(3) Chose an operator (=, +, -, x, /). This will be applied to the area selected in the next steps.

(4) Give a value (operand) fitting your plan.

(5) Click “Element Select” or “Runner Segment”.

⇒ The window: “Wall Temperature“ disappears.

(6) Select the elements using the selection tools listed in the table SelectionTools or segments using CTRL+left mouse button.

(7) You can swap or copy temperatures by pushing the arrow buttons. Hold the mouse pointer over the buttons to see what the effect of a click will be.

If you have assigned temperatures by this method, the wall temperature mode in the process parameter window will switch to “Element-based WT for filling&cooling [°C]”.

4.10.3 Wall temperature analysis

If you have already done a wall temperature analysis or you will calculate it before the filling simulation, you can use this distribution for the simulation.

1) Select "Element-based WT for filling&cooling [°C]" from the list.
2) (optionally) Click the folder button to load an already calculated distribution.

⇒ The window “Open” appears on the screen.

3) Open the appropriate TMP0 file.

If you like to use the wall temperature mode "Element-based WT for fixed times [°C]", you have to choose at least one of the offered saving modes on the card "Snapshots" in the dialogue "Options Wall Temperature Analysis". CADMOULD® 3D-F will write additional results to the known TMP files. The new CAR files have the identifier 152. They are read by the filling and packing simulation as well as by the shrinkage and warpage simulation instead of the old temperatures for filling and cooling. The time dependent wall temperatures are shown with the wall temperature dialogue.

(*Consider please* the chapters: Wall Temperatures; Options – Wall Temperature Analysis)

4.11 Packing

(Not part of the module "CADMOULD® 3D-F FILL")

Conditions:

1. Material data include pvT data.
2. CADMOULD® 3D-F PACK license.

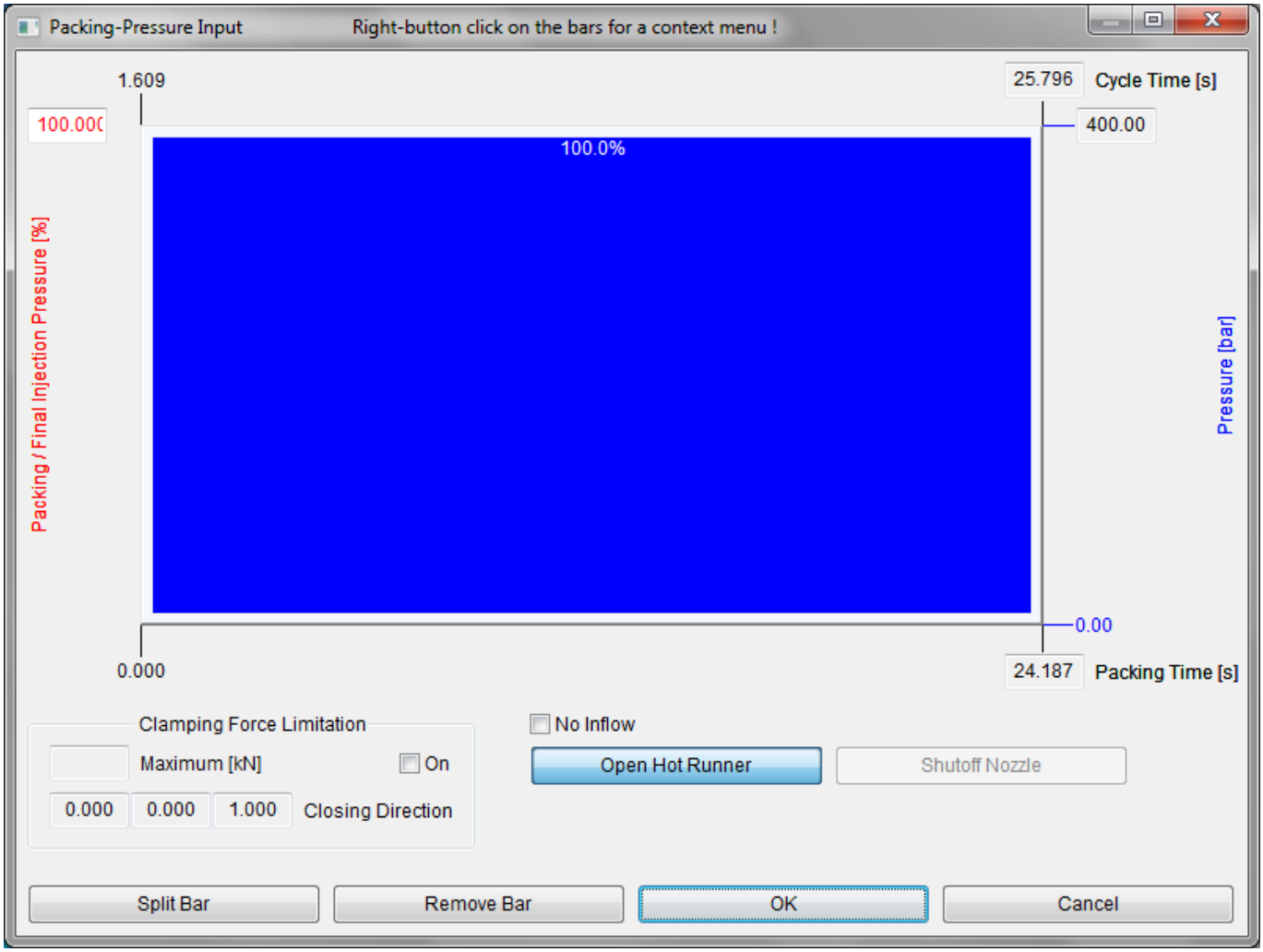
In the menu "Process Parameter" define the duration of the packing phase [s] and activate the check box "Packing Phase" (√).

The packing pressure is a constant pressure by default. The value is dependent on the material.

To change or control these settings follow the next instructions:

1) Move the cursor to the button "Duration" near to packing.

   ⇒ The button changes to "Input". Push this button and the window "Packing-Pressure Input" appears on the screen.



2) Define a packing profile similar to defining an injection profile. The only difference is that there is no flow rate input but a pressure input. It could be defined as absolute pressure values (blue bars) or relative to the required filling pressure (red bars). You can change this by right mouse button on the particular bar and than choosing "Ratio [%]" or "Pressure [bar]".

Option: Open Hot Runner / Shutoff Nozzle

A large (hot-)runner system may preserve a pressure load beyond the end of the defined packing phase. This either can force further melt into the gating area (open system) or must unload back towards the machine in case the nozzle is shut at its tip. Normally this has no significant influence. If the gating is at a location that is critical for warpage and the material is still liquid, however, this can have significant influence. In reality the hot runner is either closed by a needle valve at the downstream end or has ambient pressure at the machine end. From now on both options are simulated exactly. The choice is made via an alternative option during the “Packing-Pressure Input”.

Choices are:

Open Hot Runner = ambient pressure at the machine end of the runner

Shutoff Nozzle = zero flow rate at the tip(s) of the hot runner

4.11.1 Packing Variation

You can launch a DOE with different packing profiles. It works in the same way, as the flow-rate variation. See chapter Flow-Rate Variation.

4.12 S+W (Shrinkage and Warpage)

(Not part of the module "CADMOULD® 3D-F FILL")

The input cooling time determines the total fixation inside the mould as well as the remaining cooling phase, which follows the packing phase. The calculated temperature course will be displayed in the results as extended packing phase.

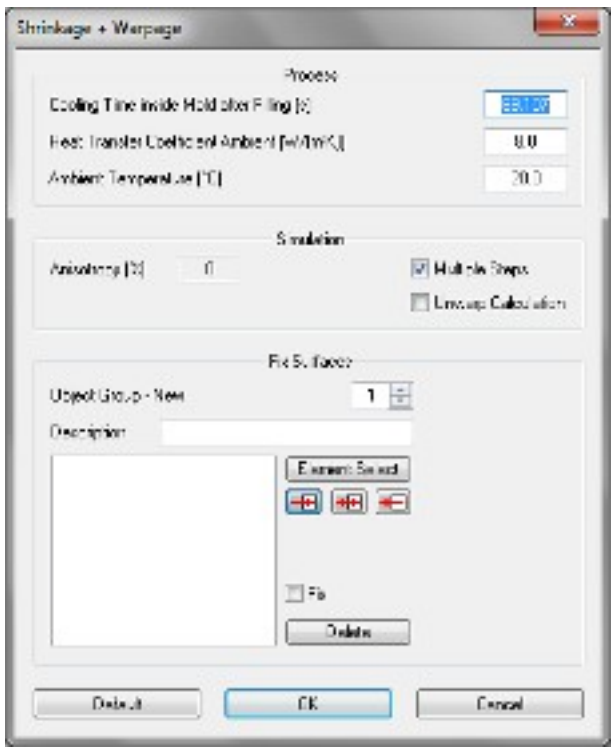
Conditions to simulate shrinkage and warpage:

1. CADMOULD® 3D-F WARP EXPERT license.
2. pvT data of the material used.
3. Mechanical data of the material used.
4. Calculated packing phase.

Typically the submenu below S+W "Input" needs no changes. Therefore it is sufficient to enter the duration value in the input window beside the button and activate the checkmark.

1) Activate the check box "S+W".
2) Enter the cooling time inside mold after filling.
3) Place the cursor on the button: "Duration [s]" near to "S+W".
   ⇒ The button changes to "Input".
4) Click this button.
   ⇒ The window "Shrinkage + Warpage" appears on the screen.

Enter the shrinkage and warpage process parameter:



- ♦ Cooling Time inside mold after filling [s]
- ♦ Heat Transfer Coefficient Ambient [W/m²/K]
- ♦ Ambient Temperature [°C]

  In the field "Simulation" of the window "Shrinkage and Warpage" calculation creteria can be activated resp. deactivated:
- ♦ Multiple Steps (more exact results, in addition, a higher computing time, therefore optionally)
- ♦ Unwarp Calculation – The deformation compensated geometry is saved in a extra file. To calculate this file, it takes more time. You should only use this function, if a Unwarp-geometry is needed.
- ♦ Anisotropy [%]: The values of the last fibre orientation file, needed for S+W are multiplied with this value/100%.
  (100 % => calculation as before, 0 % => fibre orientation stochastical)

To complete the field "Fix Surfaces" please see chapter "4.11.1 Fix Surfaces".

3) Click “OK”, in order to store the shrinkage and warpage process parameters.

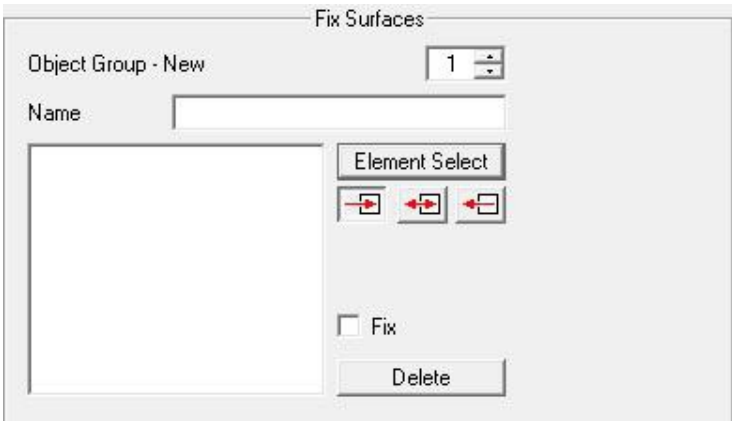
4.12.1 Fix Surfaces

In the field "Fix Surfaces" of the window "Shrinkage and Warpage" you can fix surfaces of your part. This can serve e.g. to fix the contact surfaces of the part with a steep inlay. Please consider that all these elements will be fixed in space. So spatial disconnected fixed surfaces could not move to each other.

Note: This function should generally only be used for ONE steep inlay.

Click the button "Duration/Input" next to the option "S+W" in the window "Process Parameters"

⇒ The window “Shrinkage+Warpage” with the function “Fix Surfaces” appears on the screen.



You define the surfaces which should be fixed with object groups. To define an object group, click on "Element Select".

⇒ The window “Shrinkage + Warpage” disappears from the screen.

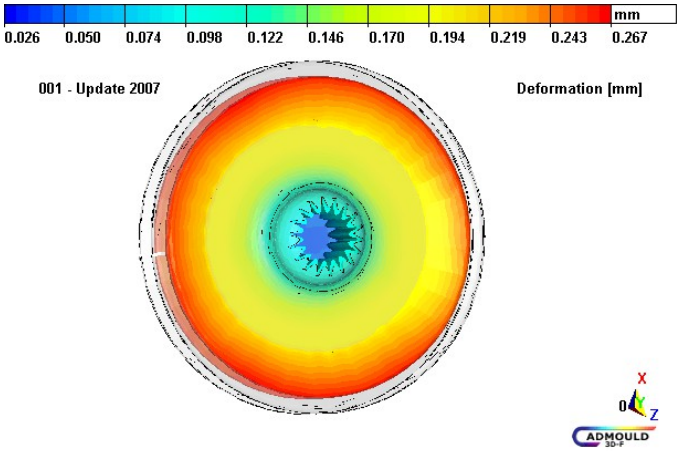
You can now add the elements you want to fix, with the selection tools to the object group.

(See also: Table Selection Tools)

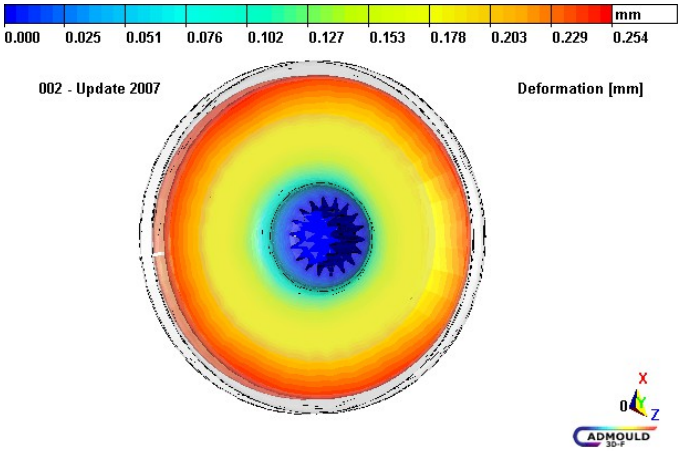
Note: You find a detailed description of object groups in the chapter: "Object Groups".

You can choose for every defined object group, if the elements in this group should be fixed by activating the option "Fix". If there are already object groups defined from other dialogs, the option "Fix" is deactivated by default.

In the following example on the next page you see a belt pulley with insert. (Pivot blanked)



Picture 1: Surfaces not fixed



Picture 2: Inner surfaces of the axle connexion are fixed

4.13 Inserts (Simulation with Inserts or 2K-Simulations)

(The function Inserts is only available, if the module “2K & Insert” is licensed.)

By pushing the button “Inserts” you get a window where you can define the parameter for inserts or first components in an insert or 2K-Simulation.

Note: To use this function it is necessary that for all parts (part as well as insert resp. first component) the cfe-mesh is already prepared.

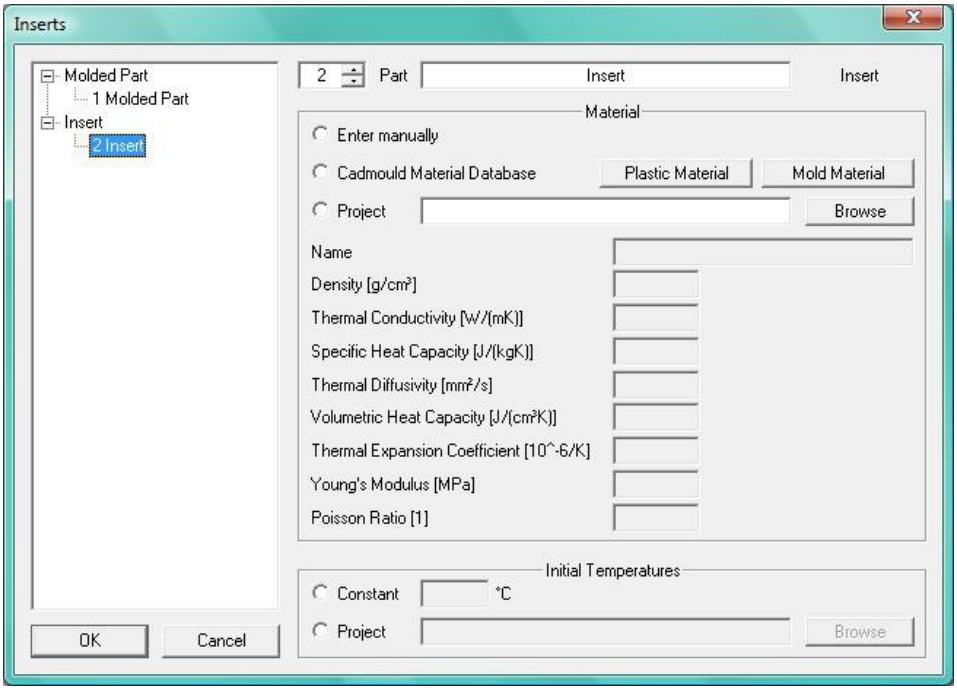
1) Push on “Inserts”.

2) The window “Inserts” appears on the screen.

3) Select “Insert”.

You can now define the required data for the insert resp. the first component.

Required date are the material and the initial temperatures, which can be defined in the following ways:



Material:

- the material data can be entered manually or chosen from the table beside,
- a plastic or mould material from the CADMOULD® 3D-F Material Database can be chosen or
- a material from a precedent project can be loaded (e.g. from the simulation of the first comp.)

Initial-Temperatures:

- define a constant temperature as starting-temperature
- load the starting-temperatures from a project, in the case the insert was also calculated with CADMOULD® 3D-F before. Therefore you need the interior temperatures, this means CADMOULD® 3D-F reads the last .car file with the number 132.

If you have made these input, you can leave the window with “Ok” and the checkmark in front of “Inserts” will be activated.

Note: Please see also chapter *Option-Inserts*, as well as the Manual 2K & Insert.

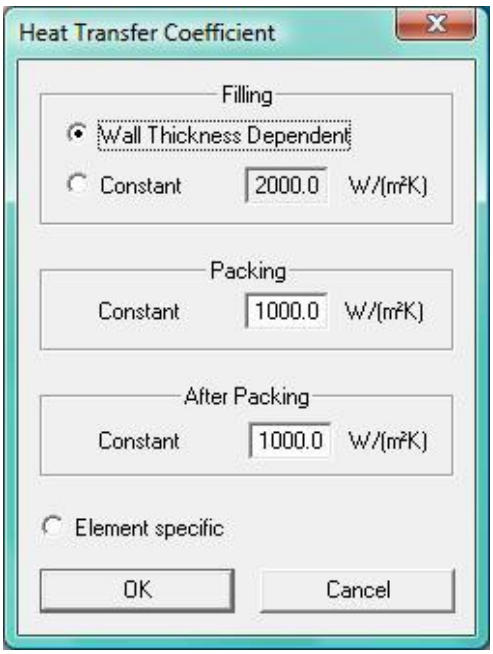
4.13.1 Boundary and Initial Conditions

- On the contact surface there is ideal adhesion.
- At the contact surface the insert and the part exchange heat.
- At all other surfaces heat is exchanged with the mould.
- The starting temperature for the insert is either constant or comes from an antecedent simulation.
- If an insert consists of fiber reinforced material, the fiber orientation is read from a previous Cadmould® 3D-F filling analysis.
- The thermal stress is calculated inside the part and the inserts.
- If a 2K simulation is calculated and a filling and packing analysis was done for the first component, the thermal strains from before the ejection can be taken over to the calculation of the second component. This happens, if in the dialog “Inserts” a temperature distribution for the first part is loaded from a former simulation. This situation is given for a plate slide or rotary mould. Because the first component is still under mould constraint and the strains must be considered.

4.14 Heat Transfer Coefficient

The heat dissipation at the plastic surface can be affected by different disturbances. Especially inserts and over moulding of other bodys resp. sheets impair the cooling at the contact faces. This affects always the cooling time, often the warpage behavior and sometimes the fill behavior.

1) Click <Options>.
   ⇒ The window “Heat Transfer Coefficient” appears on the screen.
2) Determine the heat transfer coefficient for filling, packing and after packing.



Filling phase:

(not available in CADMOULD® 3D-F RAPID)
The heat transfer coefficient can be set optionally, wall thickness dependent or be given as constant value.

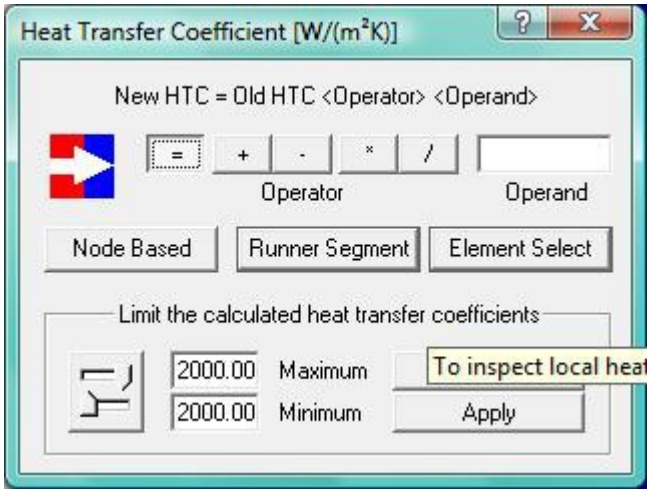
Packing and After Packing:

A constant value can be assigned to the heat transfer coefficient.

Heat Transfer Coefficient Element specific:

Activate the heat transfer coefficient

- Element specific
  ⇒ The window “Heat Transfer Coefficient [W/(m²K)]” appears on the screen.



You can now assign the heat transfer coefficient for every several element.
This heat transfer coefficient is effective for filling, packing and after packing.
To change the heat transfer coefficient, you have to choose the operator and operand first. Then you can select one of the functions "Runner Segment" or "Element Select".

Function "Runner Segment":

1) Click the button "Runner Segment".
   ⇒ The window “Heat Transfer Coefficient [W/(m²K)]” disappears from the screen.
2) Now you can choose the segments, which you would like to assign a new heat transfer coefficient with
   <Ctrl + left mouse button>.

3) To return to the window "Heat Transfer Coefficient" click right mouse button and confirm the changing.

Function "Element Select"

1) Click the button "Runner Segment".

   ⇒ The window “Heat Transfer Coefficient [W/(m²K)]” disappears from the screen.

The element selection is done with the selection tools from the table: "Selection Tools".

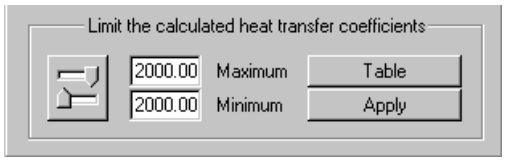
e.g.: tool: RECTANGLE

2) Hold the left mouse button pressed and describe the diagonal of a rectangle.

   ⇒ All elements which are completely included by the rectangle are selected. You are asked, if you want to change the heat transfer coefficient of these elements.

Limit the heat transfer coefficient

The calculated heat transfer coefficient can be limited by the function “Displayed Limits” and be assigned with the key "Apply".



All elements with a lower heat transfer coefficient are set to the lower interval limit. All elements with a higher heat transfer coefficient are set on the upper limit.

Heat transfer coefficient table

The button "Table" in the dialog "Heat Transfer Coefficient" opens a coefficient sorted element table. By clicking the column-head, the table will be sorted after this column. Clicking one of the elements in the table will show a blue cone on the element in the geometry.

Note: This cone can partial not be visible. To find the cone, zoom out of the part, until you see it, because the cone, different of the part doesn’t changes its size.

Clicking an element on the part surface will move the table to this element in the list. You can edit the heat transfer coefficients in this table. The option "Table" is not available in CADMOULD® 3D-F RAPID.

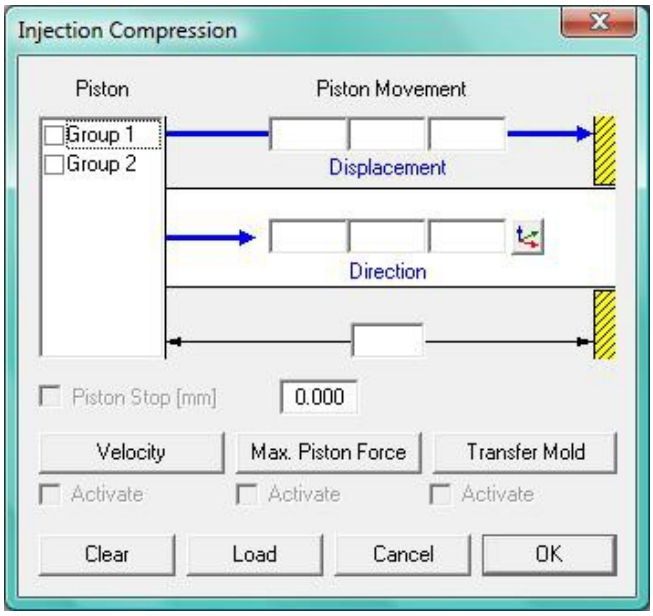
Typical Heat Transfer Coefficients [W/m²K]

Metal	1000
Plastics	200
Sheet (compact)	200
Sheet (foamed)	100
Glass	50

4.15 Injection Compression

If you want to use the process "Injection Compression" click on <Input> next to it.

⇒ The window "Injection Compression" appears on the screen.



Input all data required (eg Piston movement, Velocity and group). The at first simulated velocity phase can be divided in multiple steps. That applies also to the (optional) following force defined phase. At least either a force or travel of pistom limitation should be set.

When all inputs are made, you can activate the process and use it for the simulation. The check box is grey until you can set it.

The special process **Transfer Mold** can be found in this window as well; here, the group of the preform and its temperature at the beginning must be defined. The input of a filling time in the process parameters is necessary, but is overridden by the definition "No Inflow".

You can find new diagrams in the result selection after the simulation: Piston Travel, Piston Velocity and Piston Force.

4.16 Describe and store the simulation.

(1) Click "Simulation Description" (ID button).

⇒ The window "Simulation Description" appears on the screen.

(2) Check the path and the directory, in which the results will be stored.

a) if you want to accept the path:

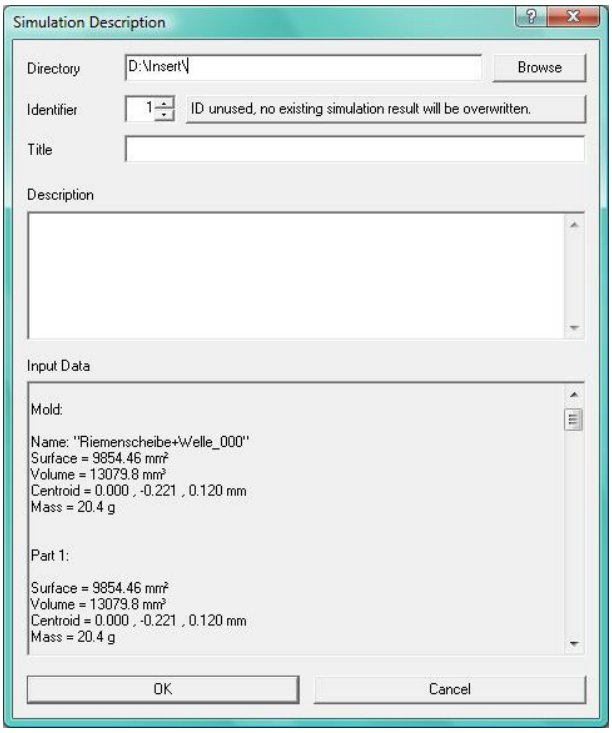
⇒ Go on with point (3).

b) If not:

1. Click "Browse".

⇒ The window: "Save as" appears.

2. Set the path and the directory.
3. Click "Save".



(3) Chose an identification number in the field "Result Identifier". (Usually simulations are given ascending ID numbers)

(4) Enter a name for the simulation run in the field "Title". This will appear with the results.

(5) (optional) Describe the simulation process in the field "Description". This is stored to a text file for later access.

(6) Click "OK".

4.17 Start the simulation

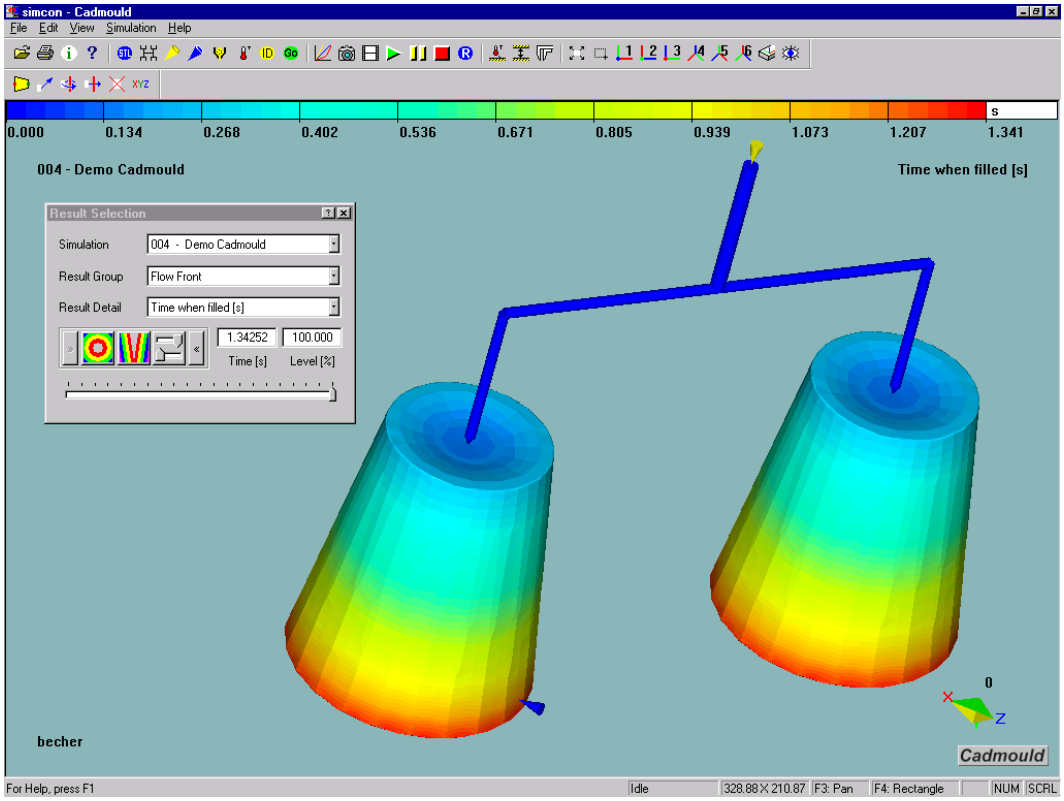
Before you start the simulation, please consider the chapter:

"Options"

(1) Click "Simulation Start" (Go button).

(2) Wait, until you get a message that the simulation is finished.

During the simulation the display shows the properties at the current filling level.

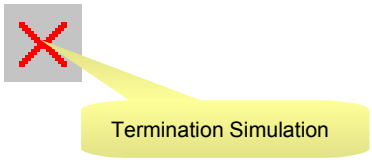


When the simulation is finished, the diagram shows the part with the properties at 100% filling. That corresponds with a filling time of 1.34252s.

4.18 Cancel the simulation

A currently running simulation can be cancelled.

After the start of a computation the button "**Start Simulation**" is turns into the new button "**Termination Simulation**".



If you click it, the calculation is cancelled at the next possible time, but earliest after the end of preparing the CFE - mesh.

4.19 Stored Results

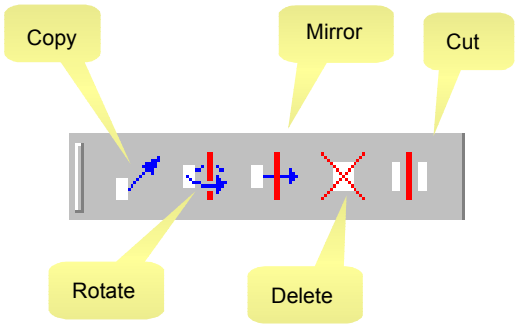
All simulation results were stored automatically in files with extensions .RM1 and .CAR in the directory you indicated or confirmed when entering the ID.

(*See* chapter: "Description")

5. Toolbar "Edit"

Activate the button: **Edit**

Pull down menu: *View/Toolbars/**Edit***



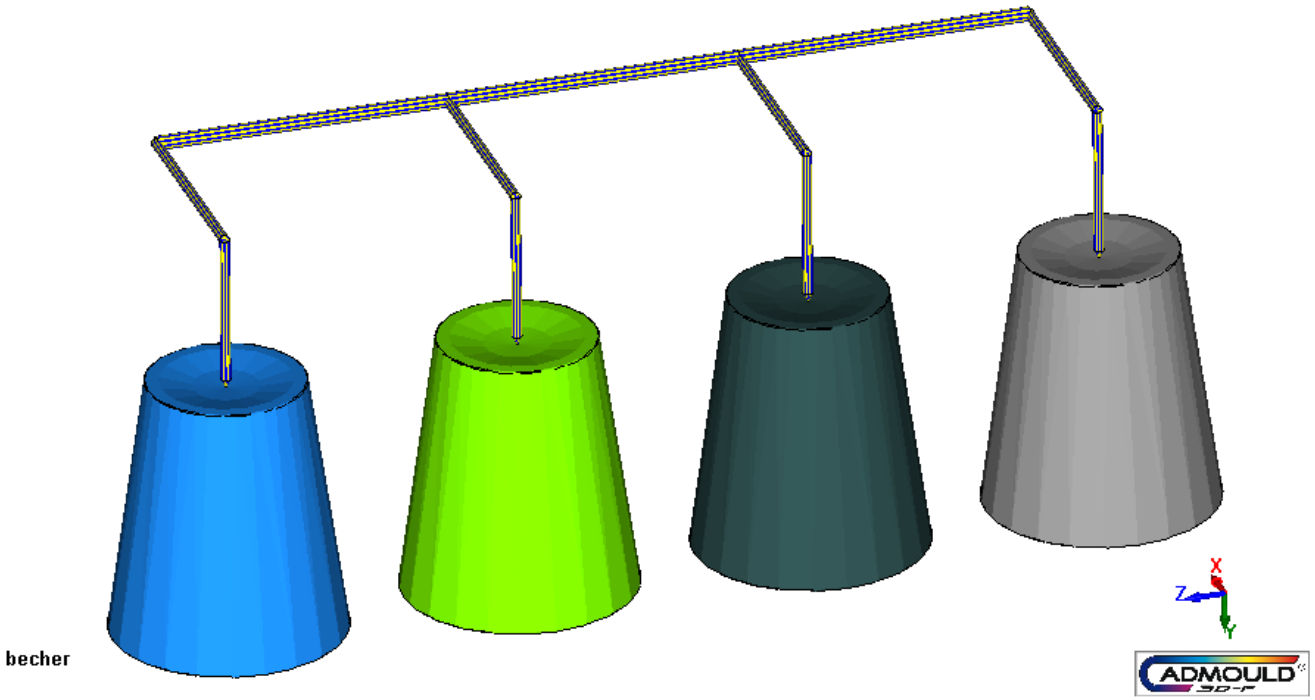
5.1 Copy

You can copy all your parts, all runner segments and all Heating/Cooling segments.

You copy the parts by marking them, entering the direction and the amount of intermediate copies.

Take a look at the following example.

That is the goal.



5.1.1 Copy the part and the runner system

All coordinates can be defined by a mouse-click.

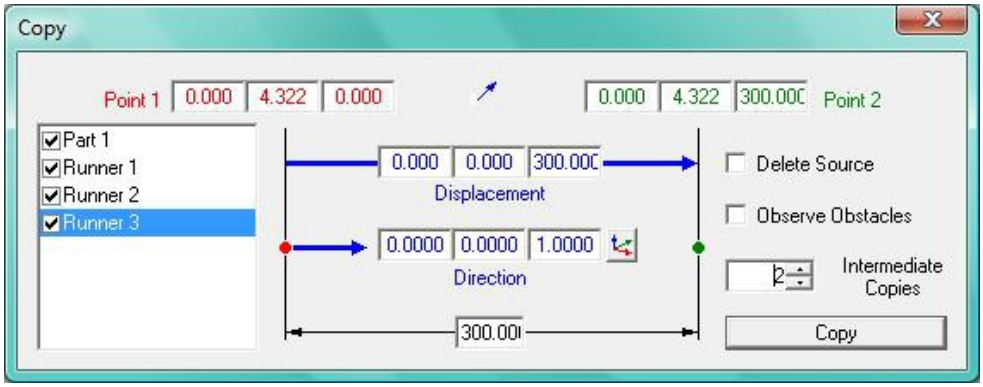
(*See* chapter*:* "Runner segment")

1) Click the button "Copy".

⇒ The window "Copy" appears on the screen.

2) Mark the parts you want to copy.

⇒ The parts you want to copy are highlighted red on the screen.



(Part 1, Runner 1, Runner 2, Runner 3)

You have to enter the displacement now. You can do this by:

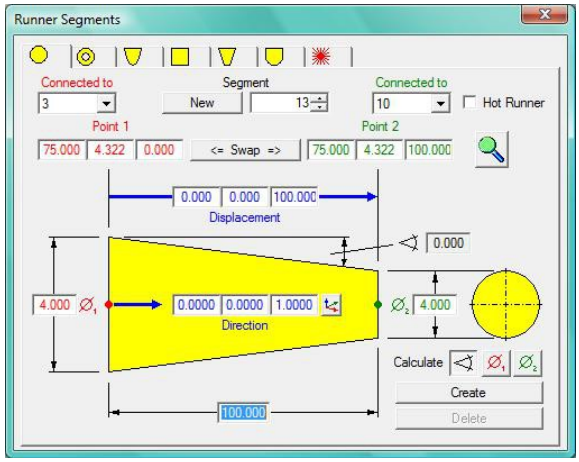
- ♦ entering two points (<Point 1> <Point 2>) or
- ♦ entering the direction and the length.

3) Leave the field by clicking <Tab> key.

4) Click on <Copy>.

5.1.2 Connecting all runner systems

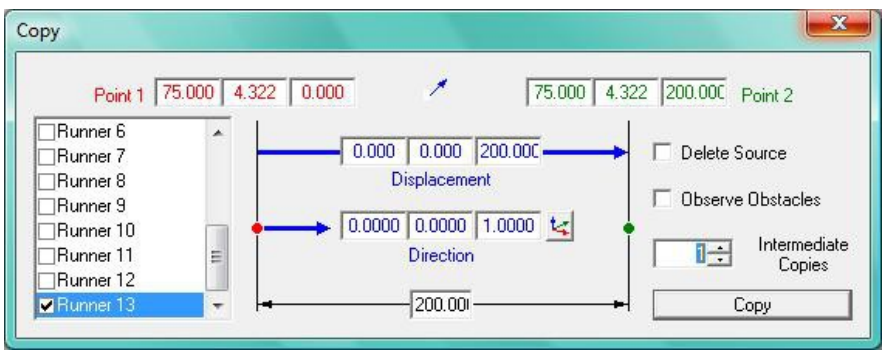
We create a runner segment connecting two runner systems and copy it to connect all the runner systems.



Open the function copy.

Mark the runner segment you want to copy, enter the displacement by selecting “Point 1” and “Point 2”.

Enter the intermediate copies.

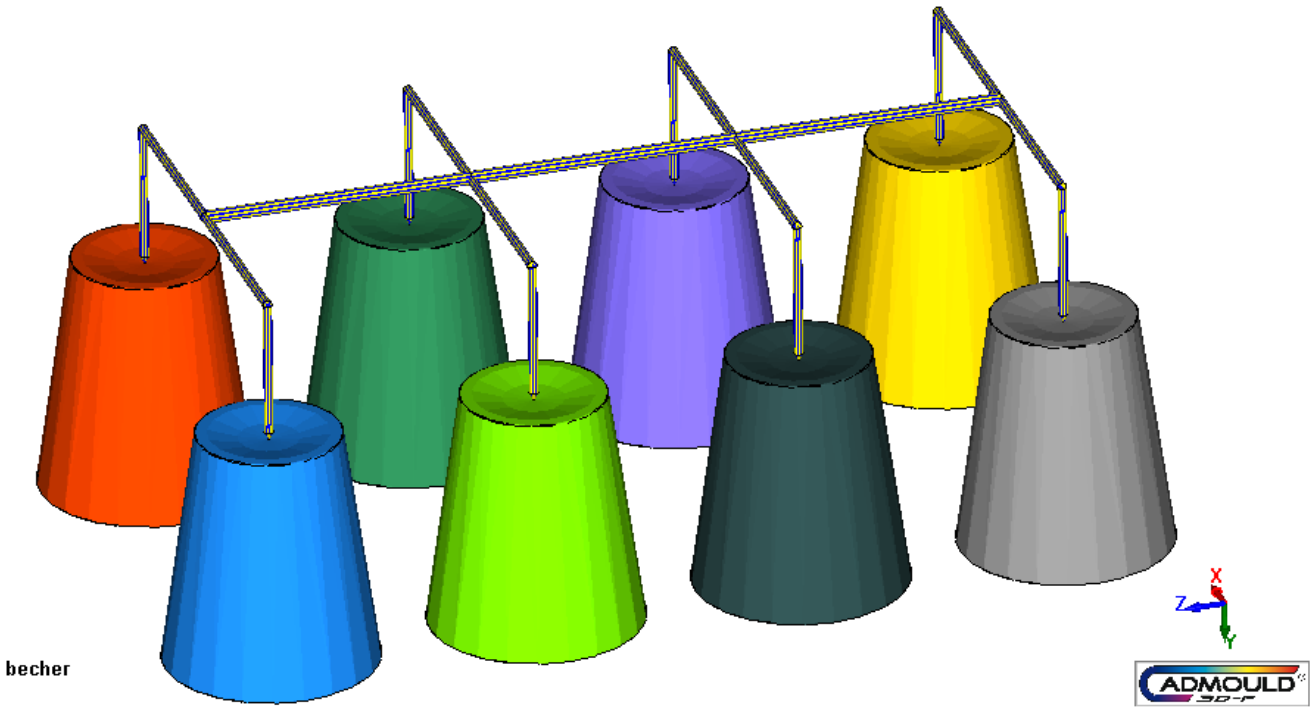


The goal is reached.

5.2 Mirror

You can mirror all your parts, all runner segments and all heating/cooling segments.
You mirror the parts by marking them, entering a point on the mirror plane and the normal vector of the mirror plane.

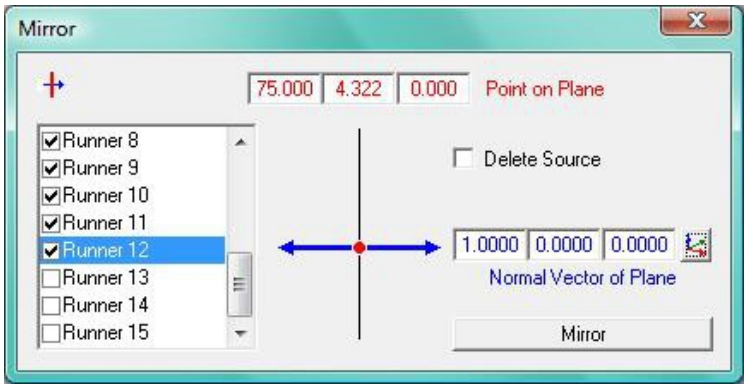
Take a look at the following example.
This is the goal.



All coordinates can be defined by a mouse-click.
(*See* chapter: "Runner segment")

In chapter: **"Copy"** we have created a runner system.
Now, we mirror the runner system in part. Mind not to mirror those segments on the backbone of the runner system!

1) Click on "Mirror".
   ⇒ The window "Mirror" appears on the screen.
2) Mark the parts you want to mirror.
3) Enter an orientation point (<Point on Plane>)
4) Enter the normal vector of the mirror plane.
5) Leave the field by clicking <Tab> key.
6) Click <Mirror>.



The goal is reached.

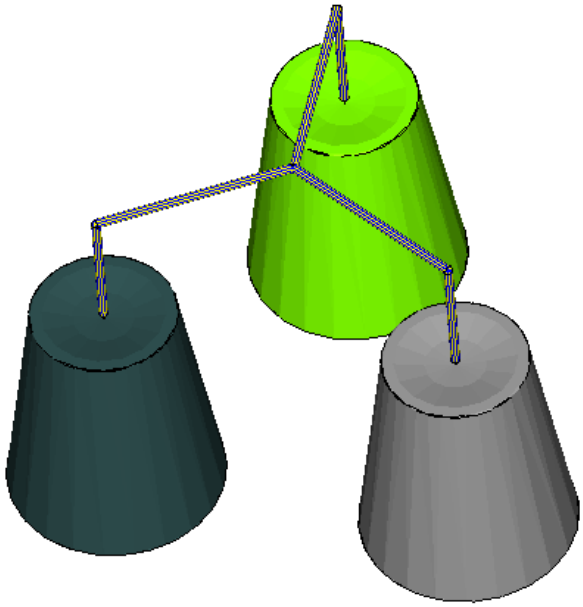
5.3 Rotate

You can copy all your parts, runner segments and heating/cooling segments around an axis.

You copy the parts by marking them, entering the axis, the angle and the intermediate copies.

Take a look at the following example.

This is the goal.



becher

Again we start with the situation finished in chapter

"Second runner segment"

All coordinates can be defined by a mouse-click.

(*See* chapter: "Runner segment")

1) Click the button <Rotate>.

   ⇒ The window "Rotate" appears on the screen.

2) Mark the parts you want to rotate, in this case all.

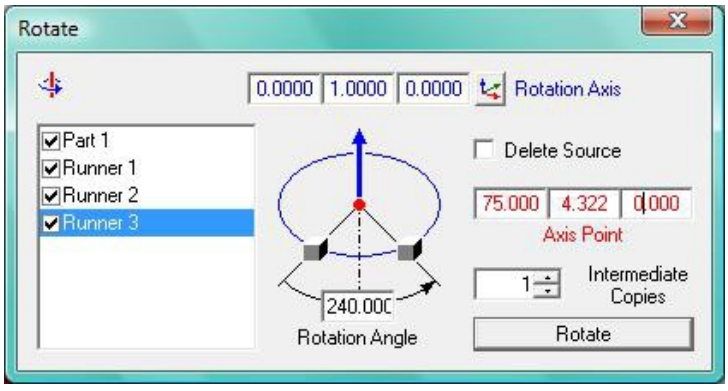
3) Enter a rotation axis by entering the axis direction and an axis point. The direction can be taken from runner 2, the point may be defined by clicking on the end of the third runner segment.

4) Enter the rotation angle. Mind that 360° would put the final copy in the same place as the source!

5) Enter the intermediate copies.

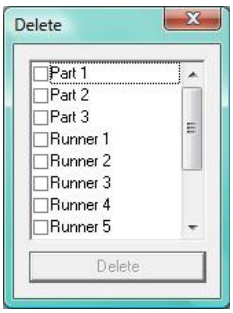
6) Leave the field by clicking the <Tab> key.

7) Click <Rotate>.



The goal is reached.

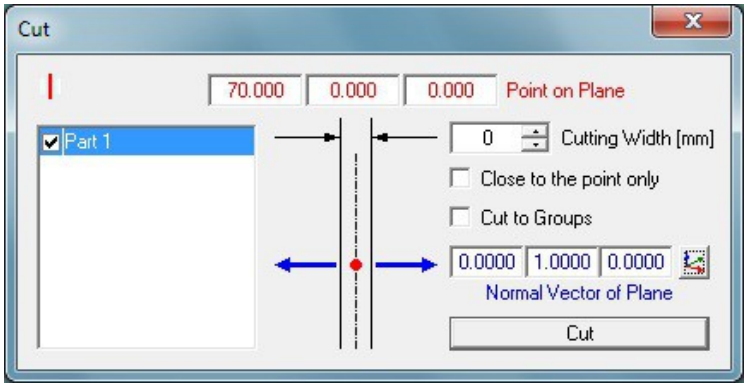
5.4 Delete



1) Click the button <Delete>.
   ⇒ The window “Delete” appears on the screen.
2) Mark the parts or runners you want to delete.
3) Click <Delete>.

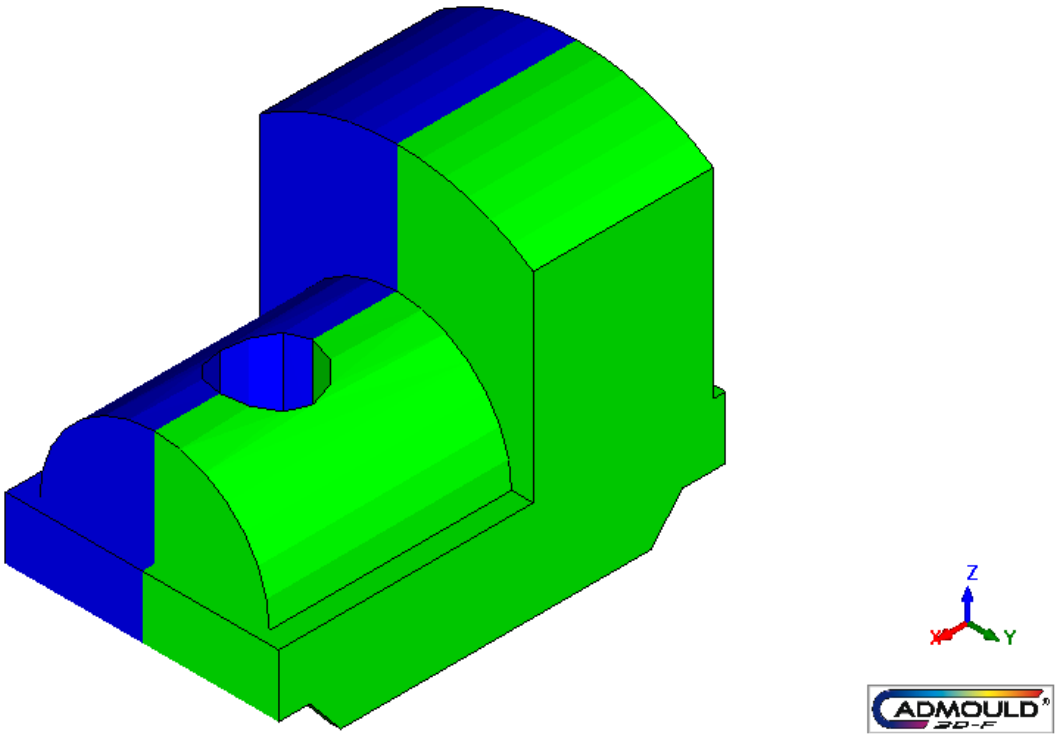
5.5 Cut

With this function, models with a known symmetry could be halved. This makes it possible to accelerate the simulation, with using symmetries, because only a half of the problem must be calculated. There is no loss of presentation, because of the symmetrical definition, the whole part is displayed.



1) Click on <Cut>!
   ⇒ The Window „Cut“ appears on the screen.
2) Mark the parts you want to cut.
3) Define the cutting plane by entering a point on the cutting plane and the normal vector to the plane.
4) Click on <Cut>!

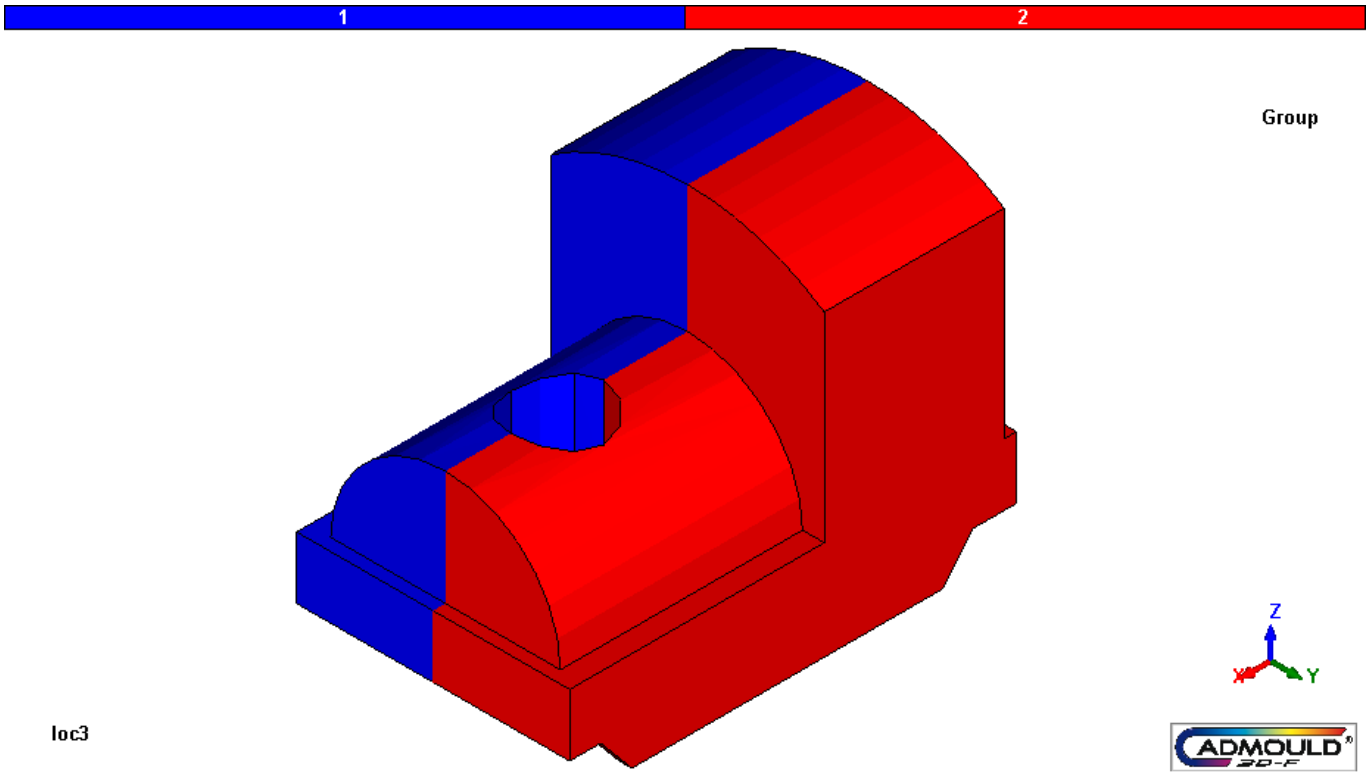
Example: Cut into parts



After cutting, the parts can be edited separative, e.g. one half can be deleted and the other can be duplicated with symmetrical definition.

Additionally you have the possibility to cut the part into groups instead of separate parts. This can be used for defining the thickness locally.

Example: Part cut into groups



Close to the points only / Cutting width

Additional you have the options “Close to the points only” and “Cutting width”. These functions help you to edit cooling systems you load as STL file and you want to convert to a Cadmould 3D-F cooling system. You can divide connected channels where in the cavity a channel is plugged. For this choose a “Cutting width” and the option “Close to the points only” to cut only through the channel you choose by Ctrl and left mouse button and not all channels crossing the cutting plane.

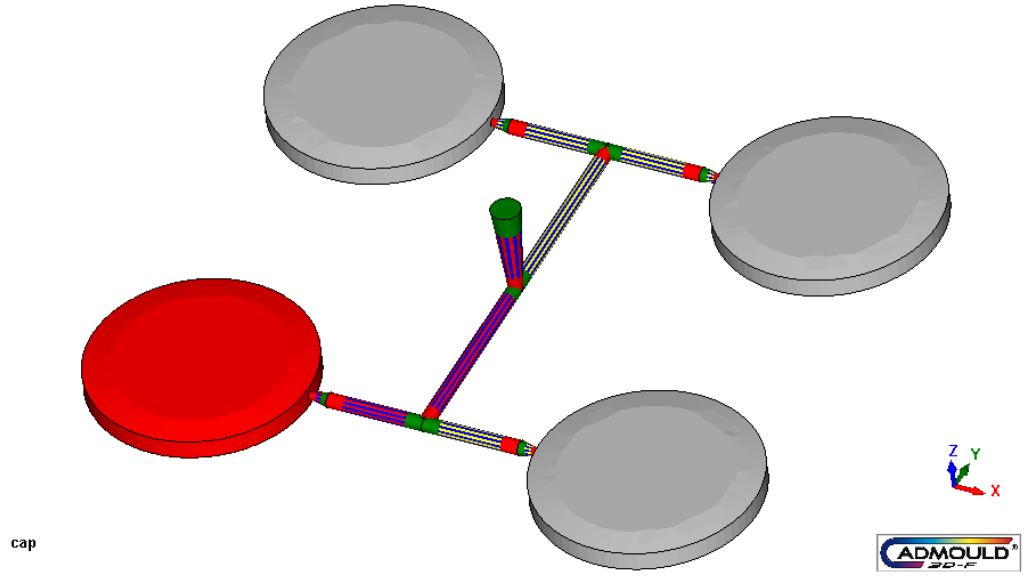
5.6 Undo – Function

The “Undo Function” makes it possible, to go back to the last state. This function can be used in the masks **Delete, Rotate, Mirror, Copy, Cut and Type Conversion** by pressing the <**Undo**> button. You can only go back one step.

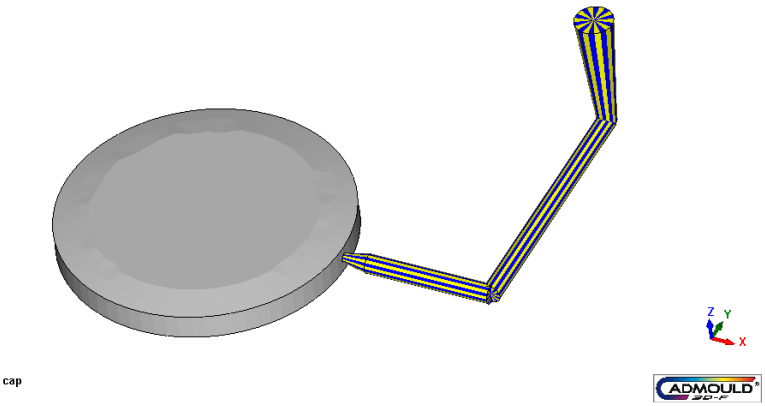
6. Design of a symmetrical cavity

In this chapter we show you the design of the following symmetrical multi cavity with the function "Build symmetrical cavity".

Therefore, the runner system with cavity marked in the picture has to be designed yet:

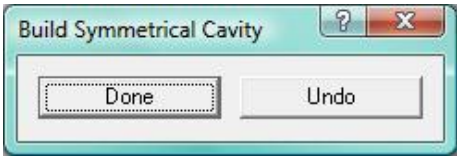


We have now the following initial situation:



1) Click in the menu Edit / Geometry / Symmetry on „Build symmetrical cavity“.

   ⇒ The window "Build Symmetrical Cavity" appears on the screen.



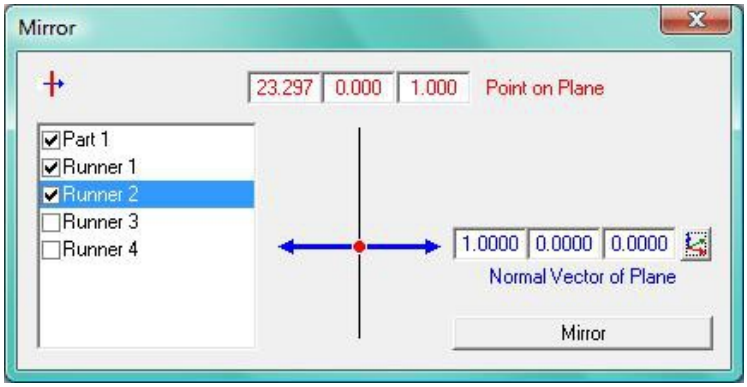
Note: This window shows, that you are in a special mode to build up a symmetrical cavity and has to be opened during the whole design process. In this mode, there are only the functions "mirror", "rotate" and "delete" from the "edit" function bar available. You finish the design process with the button "Done" and return to the normal mode.

2) Click the button "mirror" in the function bar.

   ⇒ The window "Mirror" appears on the screen.

3) Choose now the elements you want to mirror, define a point on the mirror plane and the normal vector of the plane.

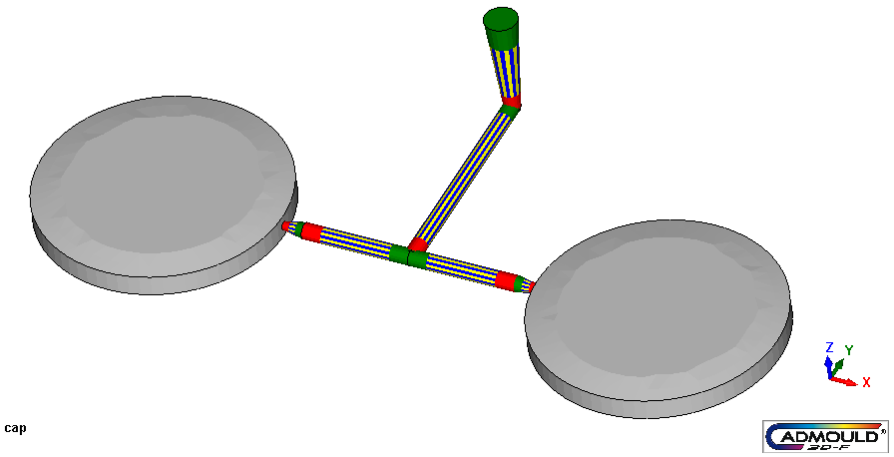
Mirror plane: As mirror plane we choose the plane spanned by the axes Y and Z. The normal vector of this plane is: X=1; Y=0; Z=0



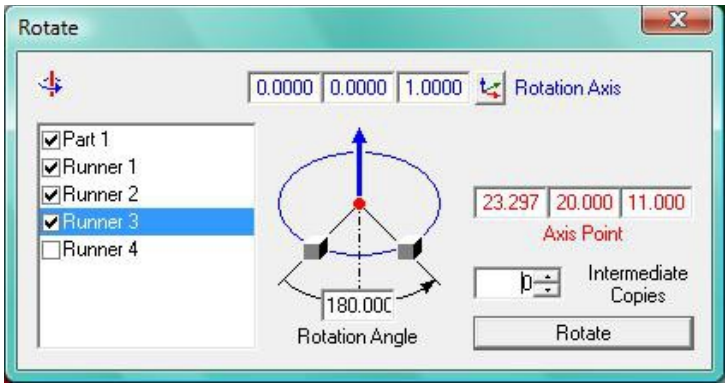
Point on Plane:: As a point on the plane we choose the final point of the third runner segment: X=23.299; Y=0; Z=1

4) Click on "Mirror".

You can now see two symmetrical parts on the screen.



5) Close the window "Mirror".
6) Click the button "Rotate" in the function bar.
   ⇒ The window "Rotate" appears on the screen.
7) Choose the elements you want to rotate, define the rotation axis, a point on the axis and the rotation angle.



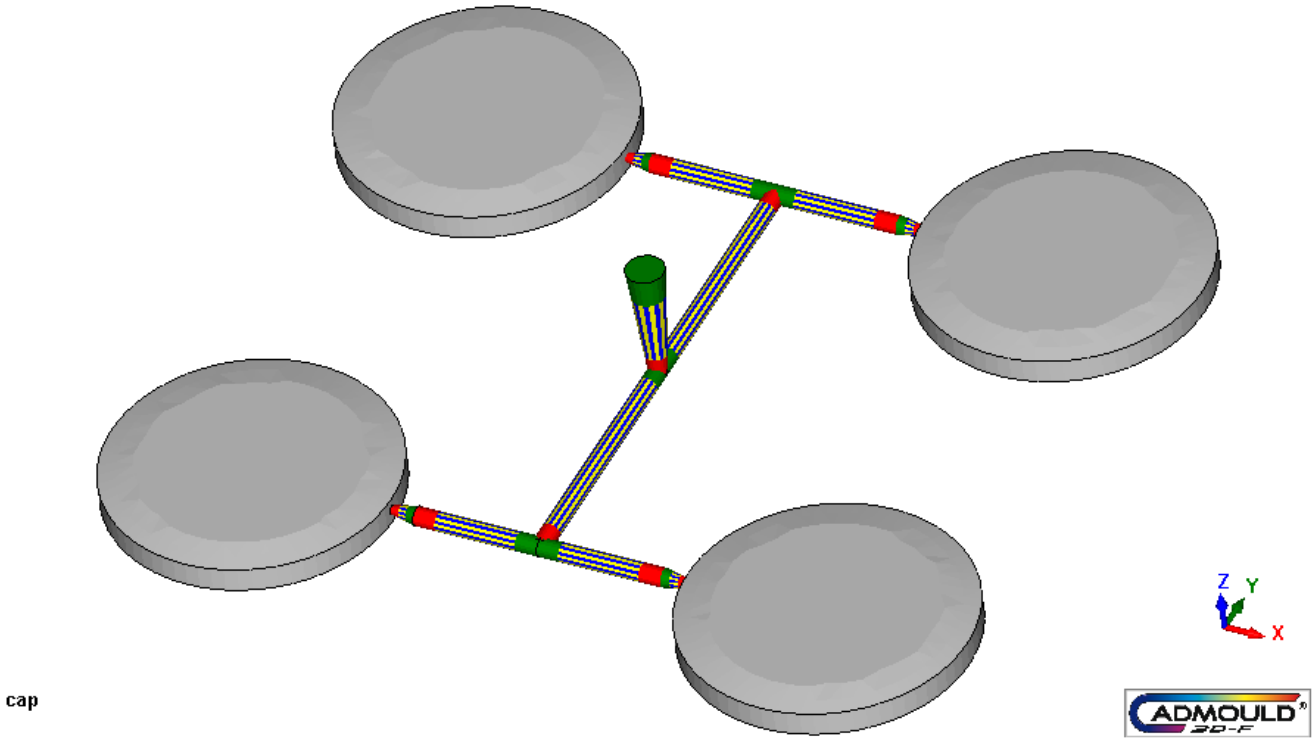
Rotation axis: As rotation axis we define the vector of the fourth runner segment. We choose it by clicking the initial or final point of the segment with <Ctrl + left mouse button>, while the cursor is set into one of the fields for the rotation axis.

Axis Point: As point on the rotation axis we choose the final point of the third runner segment: X=23.299; Y=15; Z=1

Rotation Angle: The rotation angle in this case is 180°.

8) Click on "Rotate".

You can now see the designed symmetrical multi cavity:



9) Close now the window "Rotate" and leave the symmetrical cavity mode with the button "Done".

Such a symmetrical design of a multi cavity decreases the calculating time significantly.

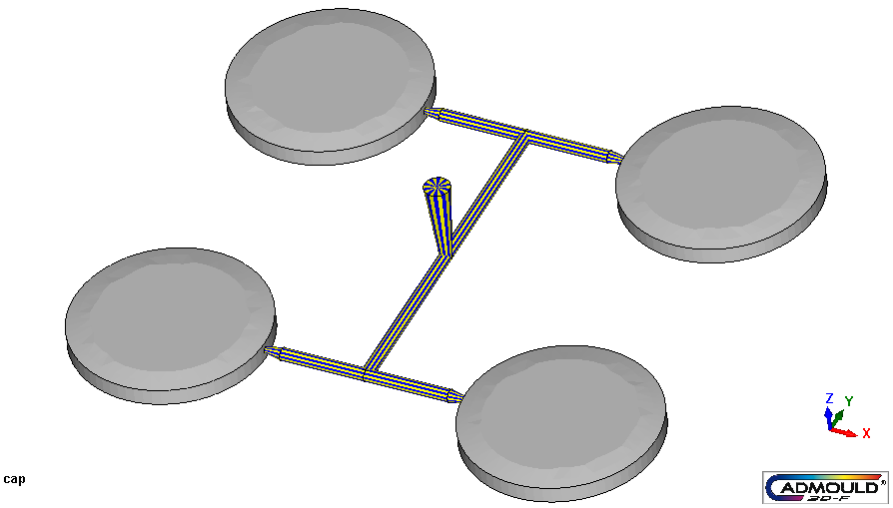
Note: You can control the design, by deactivating the option "Symmetrical objects" in the menu view / options. You can see than the situation, without symmetrical objects.

6.1 Reconstruct w/o symmetries

CADMOULD® 3D-F can also reconstruct the designed multi cavity without using symmetries.

1) Therefore click in the menu Edit / Geometry / Symmetry on "Reconstruct w/o symmetries".

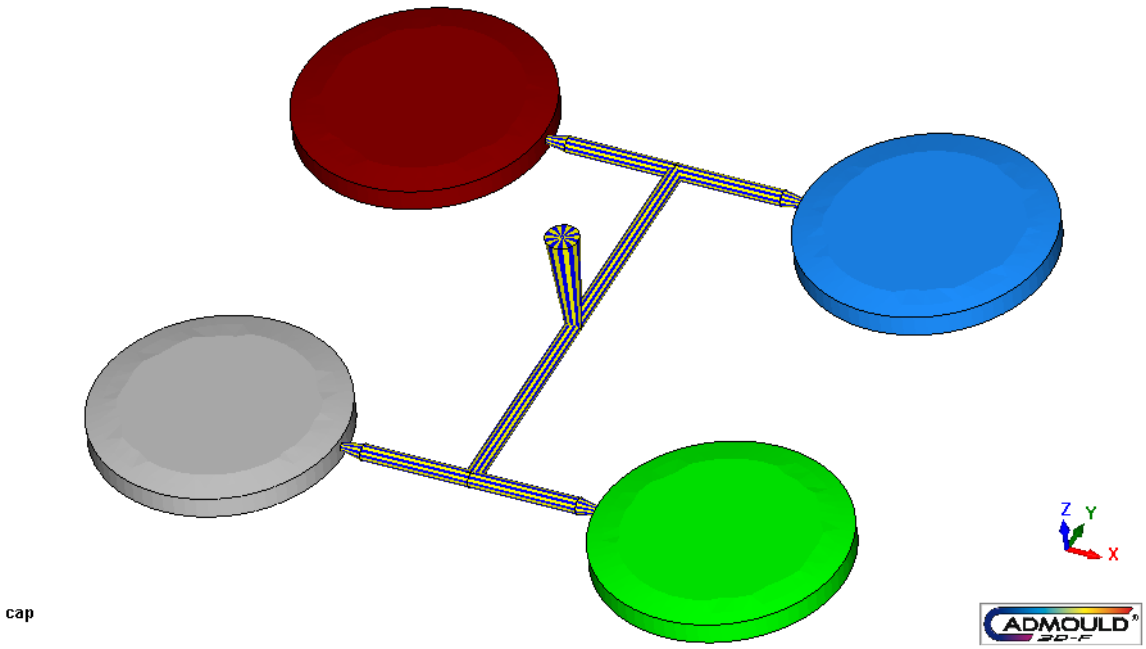
Your reconstructed multi cavity appears anymore as follows on the screen:



In your project exists now four parts and the complete runner system. Due to the fact that all parts are now calculated separately, the calculation time can increase significantly.

Note: You can clarify that your project now consists of four parts by assigning a different colour to each part in the menu View / Colours / Parts. (See also chapter: *16.3.3.4 Parts*)

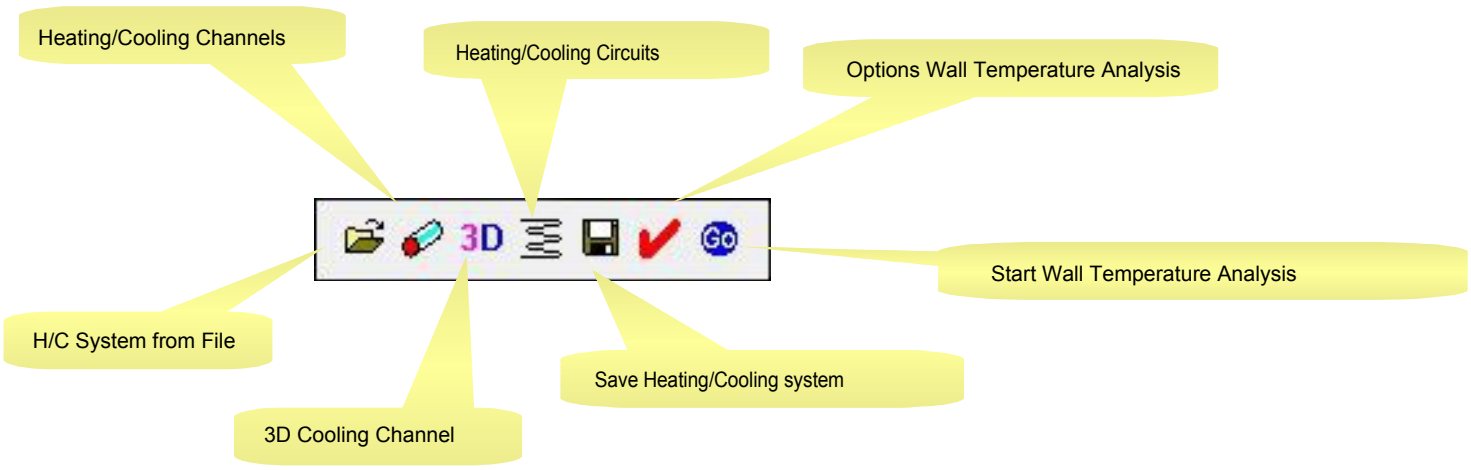
Your multi cavity appears than e.g. as follows on the screen.



7. Toolbar "Wall Temperature"

Active the menu: **Wall Temperature**

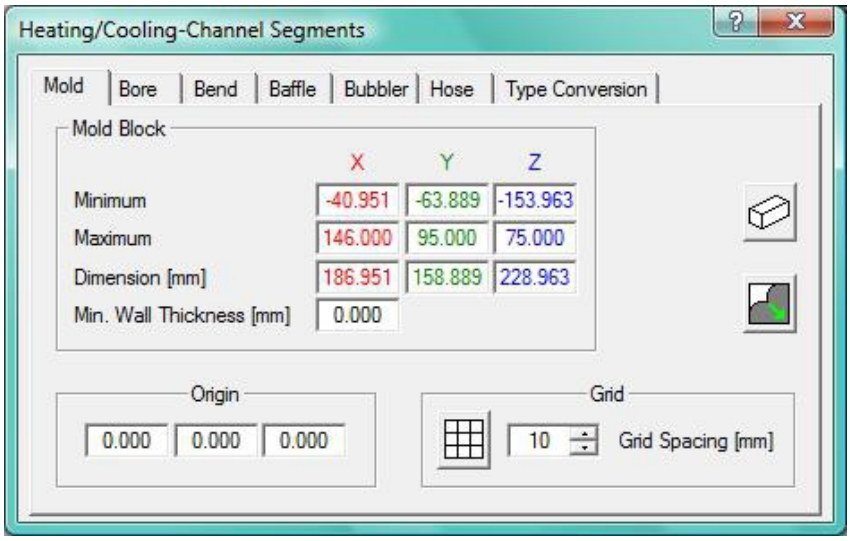
Pull down menu: *View/Toolbars/**Wall Temperature***



7.1 Mold

Click on "Heating/Cooling System".

⇒ The window "Heating/Cooling-Channel Segments" appears on the screen. You are on the register "Mold".



To support you by designing a heating/cooling system, you can now define the dimensions of your mold block. Therefore you can define the coordinates of two opposed corners or you can define a minimal wall thickness between part and mold wall.

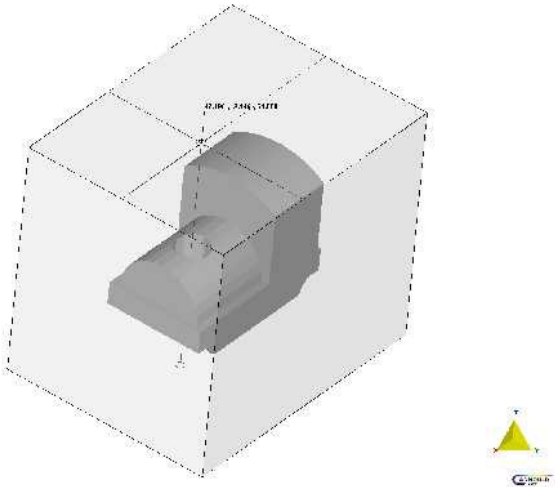
You can now show or hide the mold block with the button at the right.

Additionally you can display grid lines on the mold block and define the grid spacing.

The surface of the mold block can now be used as digitalising plane, for example to drill cooling segments into the mold.

Please see also chapter: Drilling

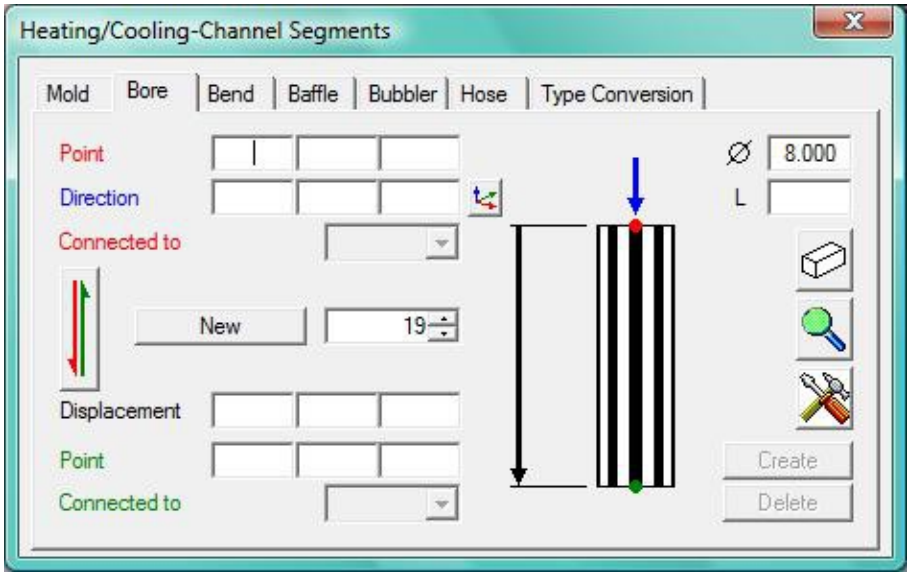
Example:



7.2 Heating/Cooling-channel segments

Click "Heating/Cooling System".

⇒ The window "Heating/Cooling-Channel Segments" appears on the screen. Jump to the register "Bore".



To input a heating/cooling segment several data are necessary. These form the information of the vertices of starting and ending point (color coded red and green) as well as the diameter. Bends and hoses are tied to existing heating/cooling-channel segment by the definition of a tangent.

There is more than one way to define these data.

Apply the method that fits your demands best. Edit fields are filled in automatically as soon as they are redundant to your inputs. E.g. defining point 1 and the displacement to point 2 makes the program fill in point 2, the length (L) and the direction.

Remark: Branches are possible.

7.2.1 Bore

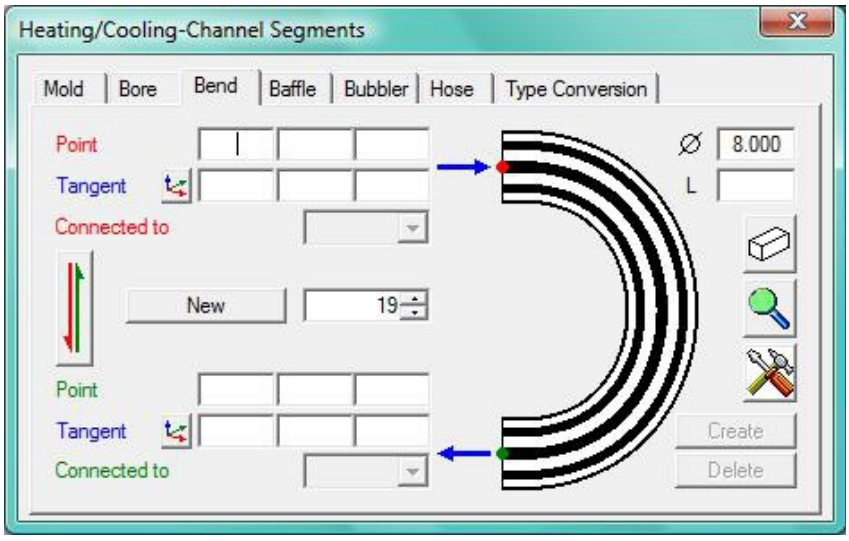
Enter the coordinates of “Point 1” (red) and “Point 2” (green), the direction, displacement, the diameter and length (L) of the bore H/C segment.

(*See* chapter: Select Function)

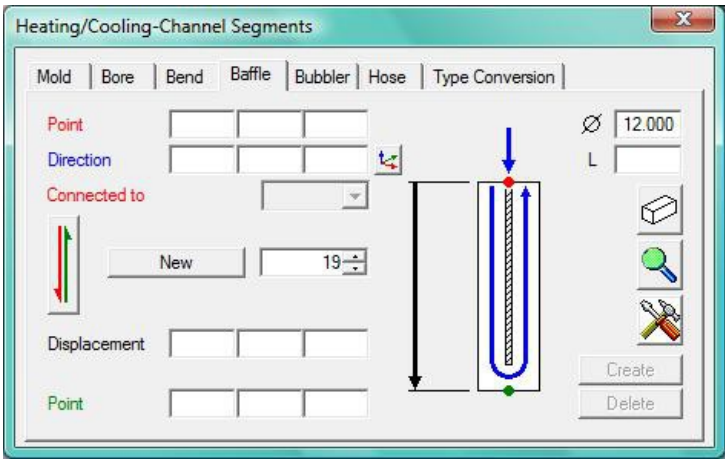
7.2.2 Bend

Enter the coordinates of “Point 1” (red) and “Point 2” (green), the tangents and the connection to, the diameter and length (L) of the bend H/C segment.

(*See* chapter: Select Function)



7.2.3 Baffle



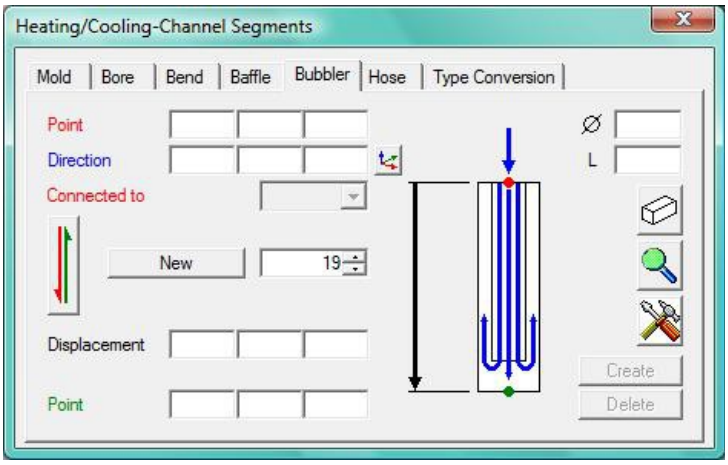
Enter the coordinates of “Point 1” (red) and “Point 2” (green), the direction, displacement, the diameter and length (L) of the baffle H/C segment.

(*See* chapter: Select Function)

Baffles are displayed striped white-purple.

Note: Baffles must be connected to two bore segments at the red end to be identified correctly by CADMOULD® 3D-F. If your baffles have the wrong direction for a correct connection, you can invert the direction manually for every baffle by clicking the button with the red and green arrow and applying with actualize. Or you can change the direction for several segments together on the card "Type Conversion".

7.2.4 Bubbler



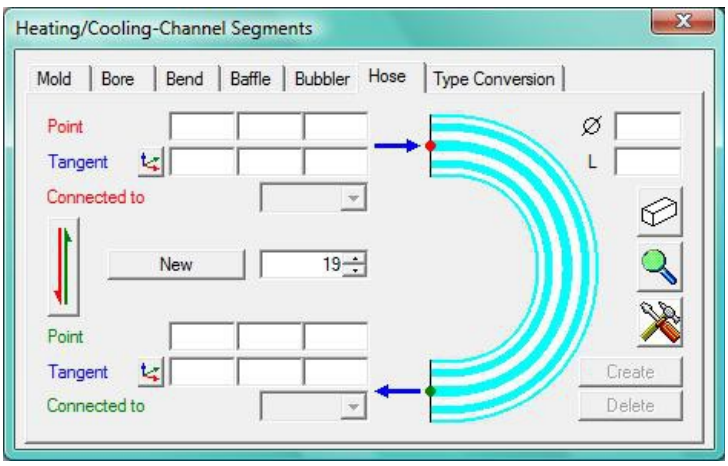
Enter the coordinates of “Point 1” (red) and “Point 2” (green), the direction, displacement, the diameter and length (L) of the bubbler H/C segment.

(*See* chapter: Select Function)

Bubblers are displayed striped white-green.

Note: Bubblers must be connected to two bore segments at the red end to be identified correctly by CADMOULD® 3D-F. If your bubblers have the wrong direction for a correct connection, you can invert the direction manually for every bubbler by clicking the button with the red and green arrow and applying with actualize. Or you can change the direction for several segments together on the card "Type Conversion".

7.2.5 Hose



Enter the coordinates of “Point 1” (red) and “Point 2” (green), the tangents and the connection to, the diameter and length (L) of the hose H/C segment.

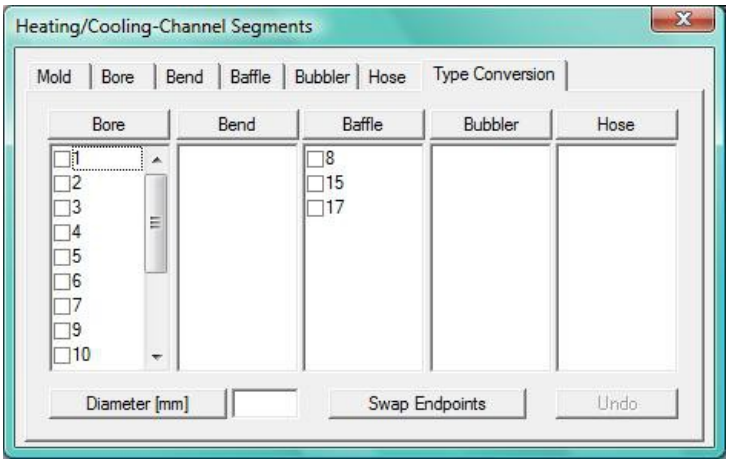
(*See* chapter: Select Function)

Hoses are displayed striped white - turquoise.

Note: Hoses are treated adiabatic. No heat transfer between other elements occurs.

7.2.6 Type Conversion

All existing heating / cooling segments are administering in the **Type Conversion** mask.



Here you have the possibility to convert all heating / cooling segments into the correct type.

A diameter can be assigned to marked heating / cooling segments.

This can be particularly helpful, if you import complete heating / cooling systems of IGES – files of axle center lines or CADMOULD® 3D-F HCS – files.

(*See* chapter: **Import Heating / Cooling systems as an IGES – file; H/C System**)

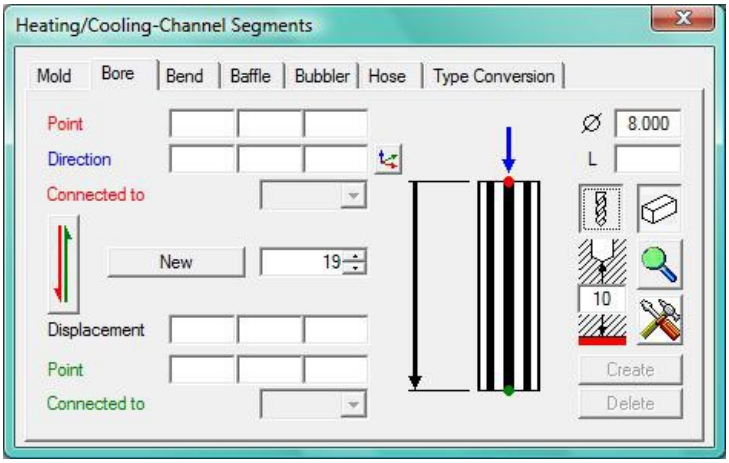
The following segment types can be converted into:

- bubbler into baffle or bore
- baffle into bubbler or bore
- bend into hose
- hose into bend

7.2.7 Design Aid for Heating/Cooling Systems

7.2.7.1 Drilling

The segments "Bore", "Baffle" and "Bubbler" can now be designed easier. Therefore, you push the button "Drilling". The register card is now displayed as follows:



In the new field you can now enter the distance between the end of the bore and the part. Now move the cursor into the field "red" point and digitize a point on the mold block surface with "Ctrl + left mouse button". From this point on will now be drilled vertically into the mold. The drilling depth (L) will be calculated automatically. You only have to enter the diameter and push "Create".

In the case, the bore don´t cut the part, it will be drilled until the opposite mold wall.

If you want to drill a bore which is not vertical to the mold wall, you can use the cutting plane function and digitize on the cutting plane.

Then the bore is vertical to the cutting plane.

7.2.7.2 Moving heating/cooling segments

It´s now possible to move heating/cooling segments by drag and drop in space. To do this, select the segment with CTRL + middle mouse button. If you select the segment on the boundary surface, you will move the whole segment. The direction in space and the length won´t change.

If you select the end of a segment, only this end will move. Take care, direction and length of the segment can change now. If two ends of segments are visibly together, they will be fixed together, if you loose the button.

After the displacement, you have to click "Actualize" to accept the changes.

7.2.8 Repair of heating/cooling segments

If you import h/c-segments from IGES-lines it´s possibly that too short segments are generated, which cause problems in the verification of the h/c-system. The function “Repair” helps you to locate these too short elements. Push the button “Repair” and the following window will open:

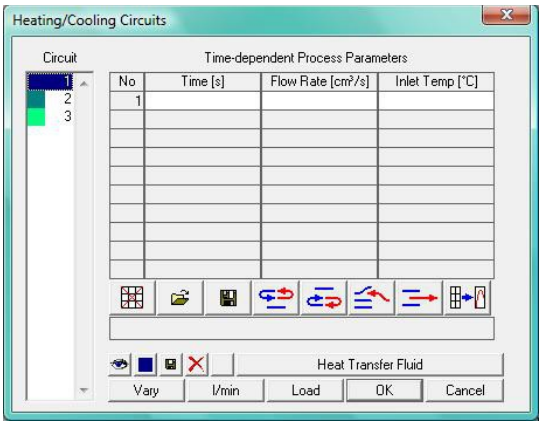
You can now enter a value for the segment length. If CADMOULD® 3D-F finds segments which are shorter you can repair them automatically by pushing “Repair” . The segments which are found are connected with lines with the “Repair”-window.

7.3 Heating/Cooling Circuits

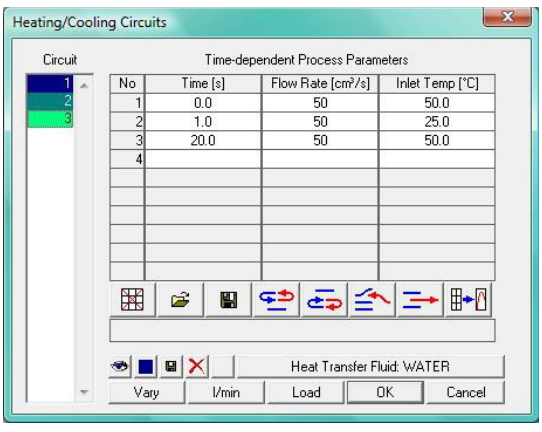
You can define time-dependent flow rate- and inlet temperature values separately for each cooling circuit. Also every cooling channel could be assigned a cooling fluid of its own.

Click "Heating/Cooling Circuits".

⇒ The window "Heating/Cooling Circuits" appears on the screen.



CADMOULD® 3D-F recognizes automatically all existing heating/cooling circuits and numbers them. You can now in the left table select one or more circuits, for which you want to define the parameters. Chosen circuits are displayed red on the graphic table.



If you have selected one or more circuits, you can now enter the points of time with the associated values for the flow rate and the inlet temperature into the table on the right side. The time you enter defines, that from this point of time until the next value resp. until the end of process, the associated values are valid. A duly completed table could for example look like this:

Now you only have to define a cooling fluid for each channel. You can do this by marking all, or separately.

The big buttons in the first row under the table serve to delete the whole table, to handle several rows of the table as well as to import/export the content of the table.

With the buttons in the next row you can show or hide the selected circuits, change the colour, save one or more circuits, delete them and change the flow direction and the cooling fluid.

The buttons in the last row serve to activate the variation of flow rate and inlet temperature, to change the unit of the flow rate, to load the data from another project as well as to accept or reject the input.

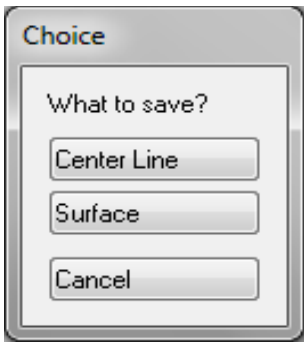
7.4 Save Heating/Cooling system

All heating/cooling circuits can be stored in a file.

1) Click “Save Heating/Cooling System”.
   ⇒ The window "Save as" appears on the screen.
2) Enter the name of the heating/cooling system.
3) Select the file format. You have the option between
   - H/C System Files (*.hcs)
   - IGES Files (*.igs)
   - Cadmould-FE-Mesh Files (*.cfe)
   - NASTRAN Files (*.nas)
   - Binary Stereolithography Files (*.stl)
   - Textual Stereolithography Files (*.stl)

   By default the *.hcs-format is selected.

   If you decide to save the system as a *.cfe-file, the choice-dialog appears



<Center Line> means, that one element with two nodes will be saved.

<Surface> will save a triangulation of the channel surfaces.

4) Click "Save".

7.5 Import a Heating/Cooling System

1) Click “H/C System from File”.
   ⇒ The window "Open" appears on the screen.
2) Mark a HCS-file and click <Open>.

H/C systems in the following file formats can be imported:

- H/C system files (*.hcs)
- IGES files (*.igs)
- Cadmesh files (*.cm4)
- CFE-mesh files
- STL files

Please see for the import of STL files chapter: 3D Heating/Cooling Channel

7.6 Options – Wall Temperature Analysis

To start the wall temperature analysis the options of the analysis must be filled in.

Input is done, if you can close the window “Options Wall Temperature Analysis” by <OK>.

Note: You can find all input data from this dialog, except of "Snapshot" in the .TXT-file of the project.

7.6.1 Material

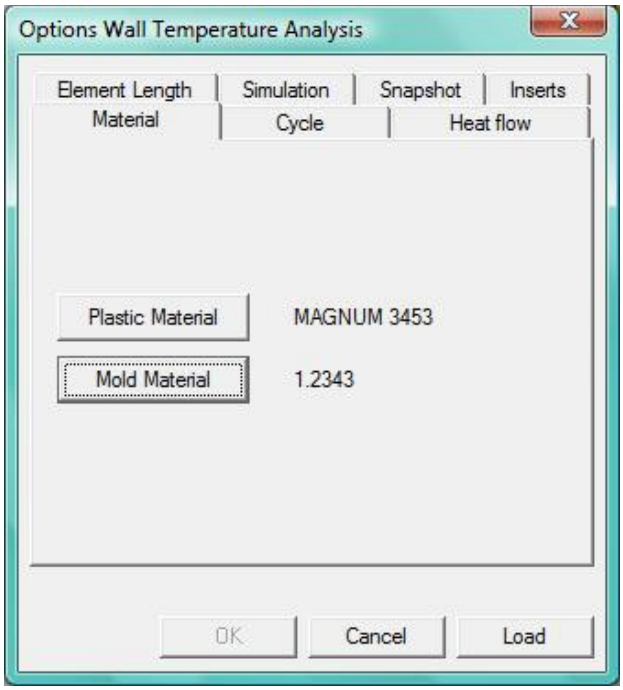
1) Click “Options Wall Temperature Analysis”.

   ⇒ The window "Options Wall Temperature Analysis" appears on the screen.

In the tab dialog: “Material” the Plastic Material and the Mold Material have to be entered.

The Plastic Material should be already registered, because you must have selected the plastic material of your part.

(*See* chapter: Wall Temperatures, Material)



Note: With the button "Load" you can optionally load the settings for the actual card or for all cards from an existing project.

2) Click <Mold Material>.

   ⇒ The material data base of metal appears on the screen.

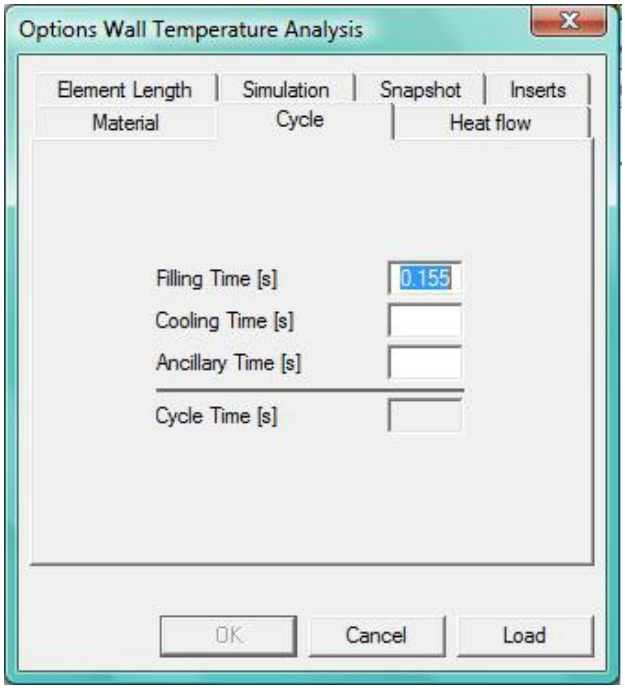
3) Search for the mold material you want to use.

4) Double-click the material name or click <Select> to finish the selection.

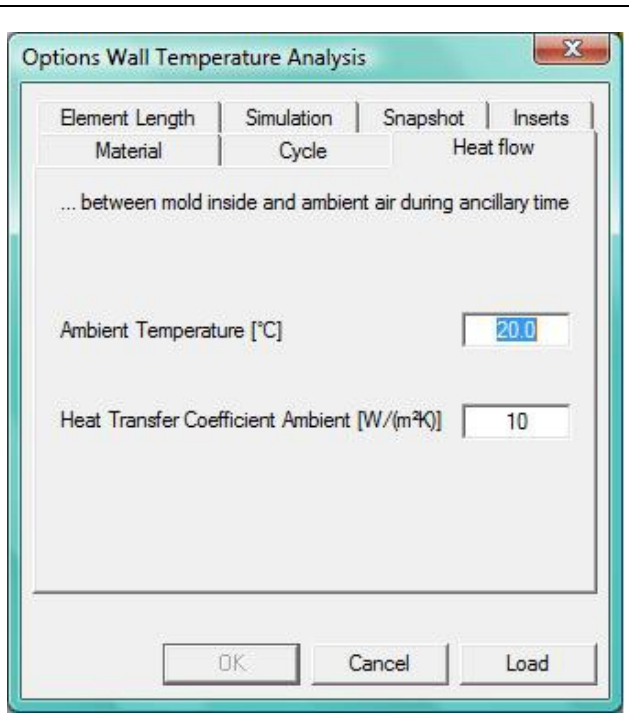
(*See* chapter: Enter the material)

7.6.2 Cycle

In the tab dialog: “Cycle” you define the cycle time of the wall temperature analysis.

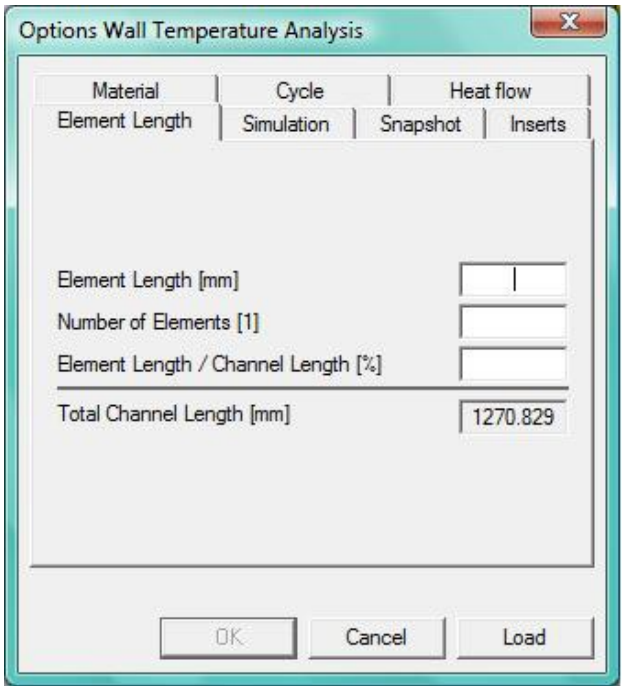


7.6.3 Heat Flow



In the tab dialog: “Heat Flow” you define the ambient temperature and the heat transfer coefficient of the ambient of the wall temperature analysis. Recommended default values are already set. These values apply to the cavity in the auxiliary processing time.

7.6.4 Element Length

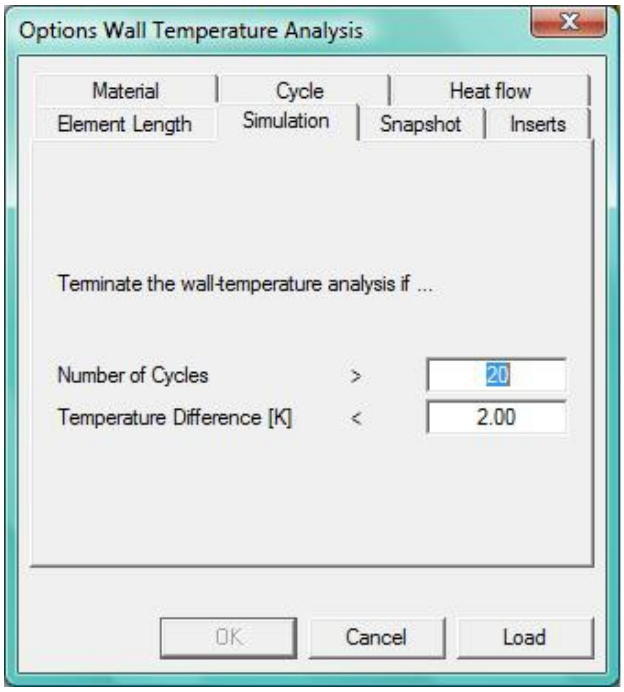


In the tab dialog: “Element Length” you define the element length of the mesh of the heating/cooling channel. The shorter the elements, the more precise the results are, and the higher the computation time. As a check the number of elements and the proportional length of the elements to the channel length is shown.

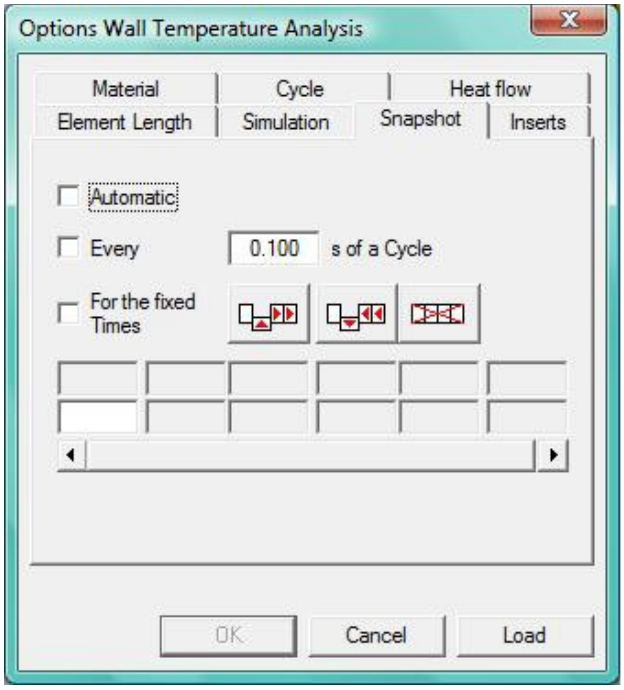
Attention: Small element lengths (analogical much elements) causes long calculating time or even memory requirements that exceed available memory.

7.6.5 Simulation

In the tab dialog: “Simulation” you define the criteria of the termination of the wall temperature analysis. The criteria are associated by an OR-condition. The temperature condition compares every location of two following cycles at the end of the cycle.

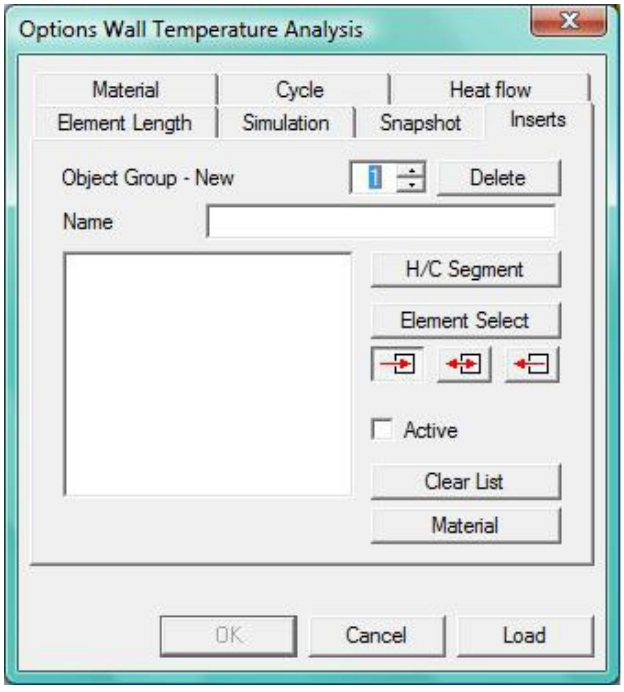


7.6.6 Snapshot



If you have chosen at least one of the offered saving modes, you can use the wall temperatures as edge condition for following simulations in the dialogue “Process Parameters”. If you choose "Every" or "For the fixed Times", this detailed results get usable by the wall temperature mode "Element-based WT for filling&cooling [°C]". CADMOULD® 3D-F will write additional results to the known TMP files. The new CAR files have the identifier 152. They are read by the filling and packing simulation as well as by the shrinkage and warpage simulation instead of the old temperatures for filling and cooling. The time dependent wall temperatures are shown with the wall temperature dialogue.

7.6.7 Inserts

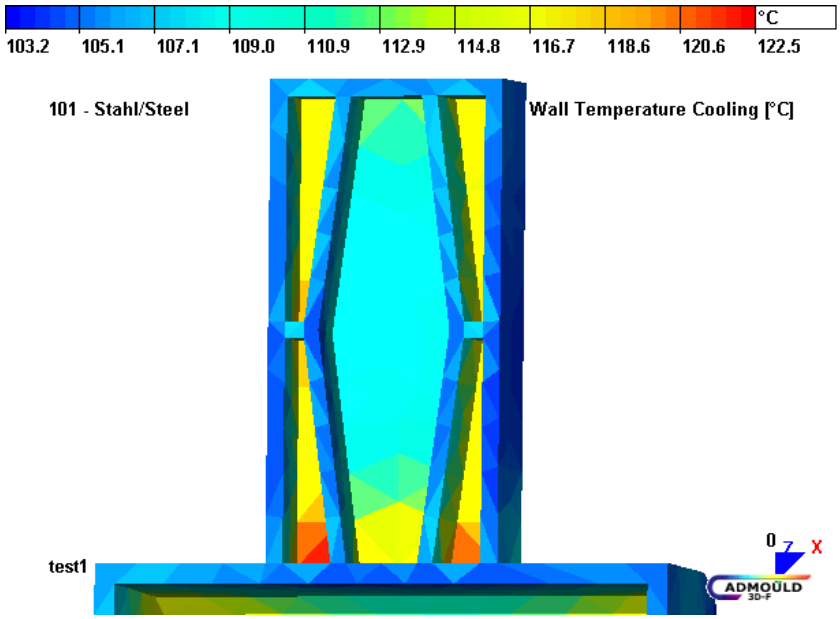


This card offers you the possibility to consider inserts from another material in your wall temperature analysis. The definition of the inserts is done with object groups, which contain the elements of another material. The object group must also contain the h/c-segments which belong to the insert. If there are no elements or no h/c-segments in the object group, you get an error message when starting the wall temperature analysis, that this insert is ignored. If you still want to consider this insert, you have to define an h/c-segment, which is nearby the insert.

In the following example you see the results of two wall temperature analyses with inserts of different materials.

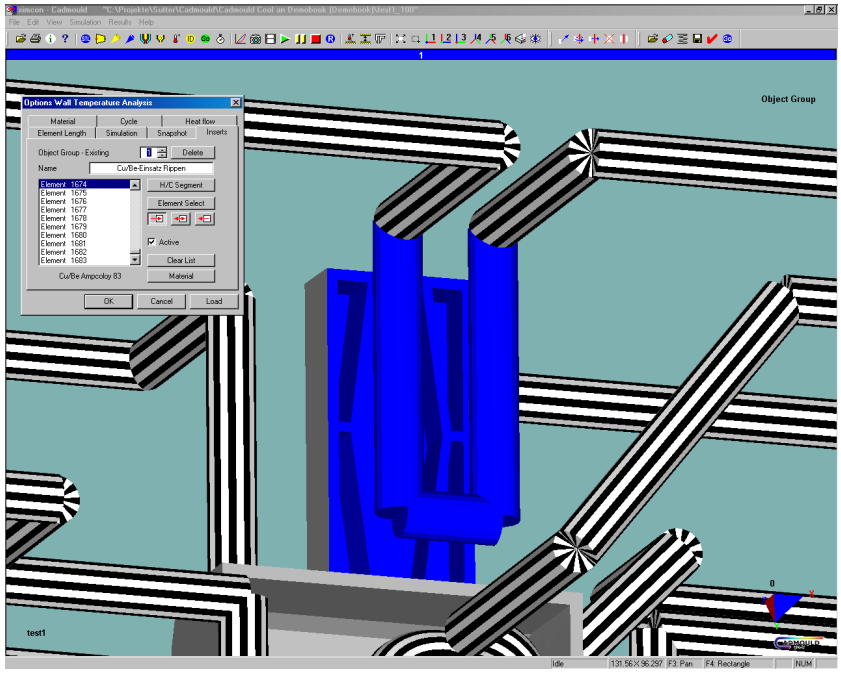
Example 1: Wall temperature analysis with a steel insert

The lowest temperature in this example is about 104°C and the Hot-Spots in the corners are about 123°C.



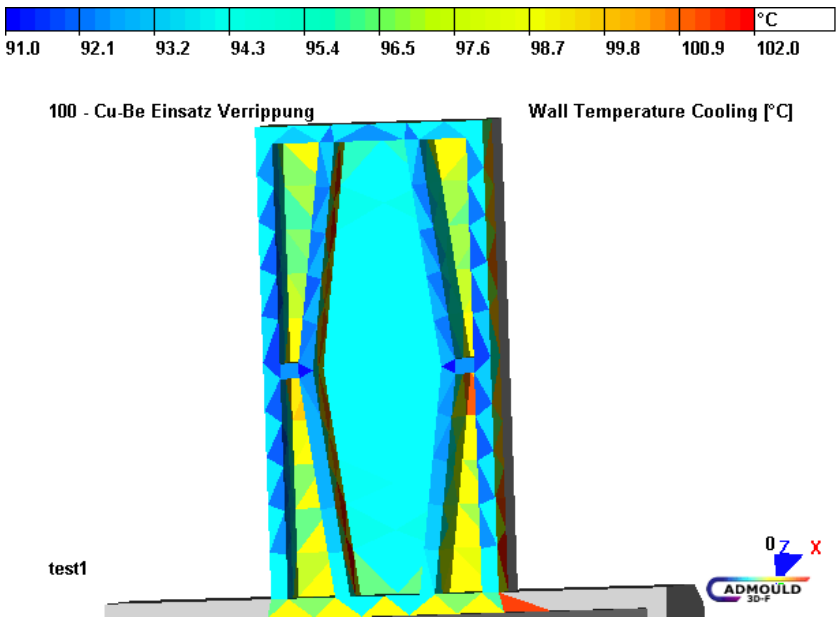
Example 2: Definition of an Cu/Be insert

At this picture, the surfaces and cooling segments which are in contact with the Cu/Be-insert are displayed blue.



Example 2: Wall temperature analysis with a Cu/Be insert

You can see a clearly temperature homogenisation caused by the Cu/Be insert at the ribbed region, as well as a temperature reduction about 25°C at the (former) Hot-Spot.



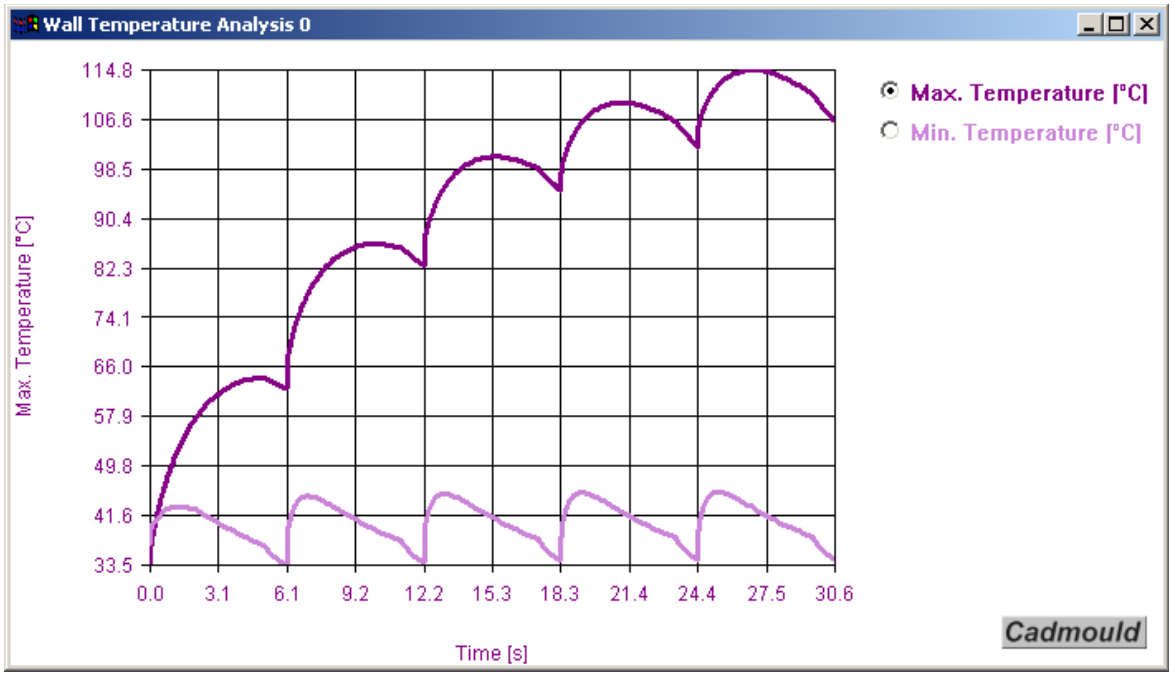
7.7 Wall Temperature Analysis

Click on “Wall Temperature Analysis (GO)”, in order to start it.

You can run only the Wall Temperature Analysis or the complete simulation depended on your process parameter settings.

During the Wall Temperature Analysis a Temperature-Time-Diagram is shown.

You get a message about the completion of the wall temperature analysis.



(*See* chapters: Wall Temperature distribution, Wall Temperatures)

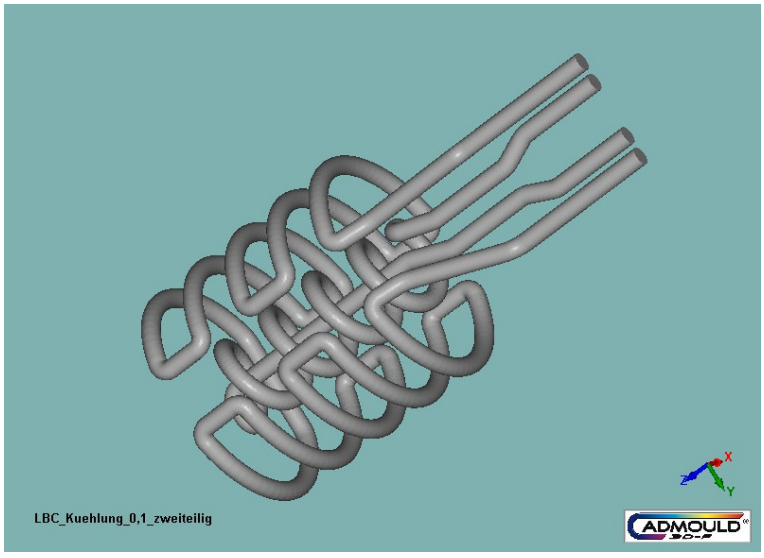
7.8 Cooling system manufactured of STL

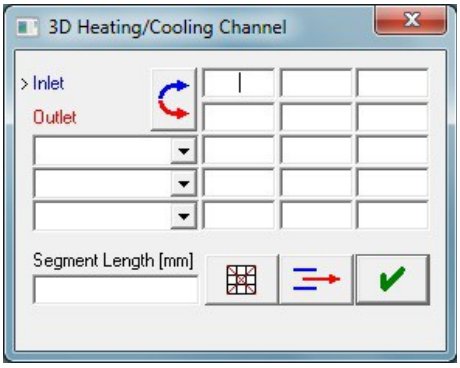
It is very simple to create a cooling channel in CADMOULD® 3D-F, if you have the cooling channel already as a STL-File. In this case there are two opportunities to move on, depending on what kind of cooling channel you have: close-to-contour or drilled channels.

7.8.1 Close-to-contour cooling channel

The latest version of Cadmould 3D-F now gives you the possibility to import heating/cooling systems from stl files and convert them automatically into the internal Cadmould 3D-F heating/cooling system format. The following example of a cooling system from ***LBC GmbH*** describes how to operate.

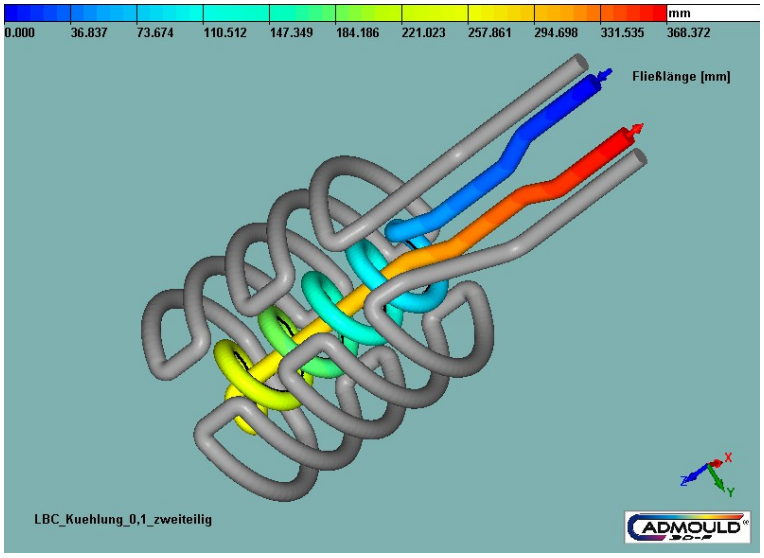
1. Click the button “Open h/c system” from the wall temperature toolbar.
   ⇒ The window “Open” appears on the screen.
2. Select “.stl” in the “Files of type” pull down menu and open the STL-file of your h/c-system.
   ⇒ Your STL and the window “3D Heating/Cooling Channel” appears on the screen.

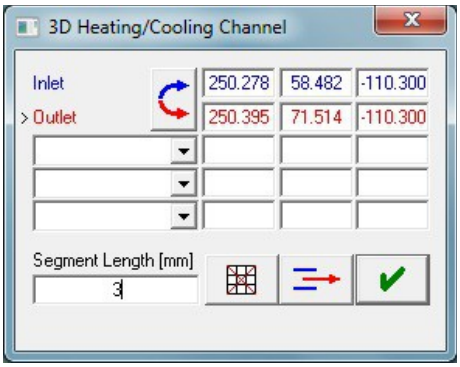




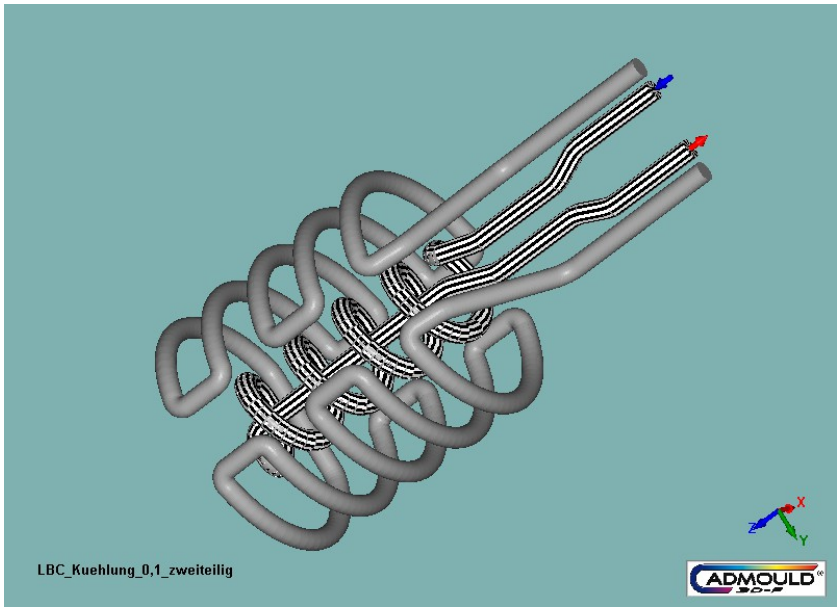
*Example: Cooling system of **LBC GmbH, Lüdenscheid Germany***

3. Digitize with CTRL+left mouse button the inlet as well as the outlet of the first h/c circuit.
4. Enter a Segment-Length [mm]. Shorter segments represents the original shape better.
   ⇒ The chosen cooling circuit is now displayed in color. The color scale shows you the flow length of the channel.





5. Click on the button with the green checkmark to start the conversion.



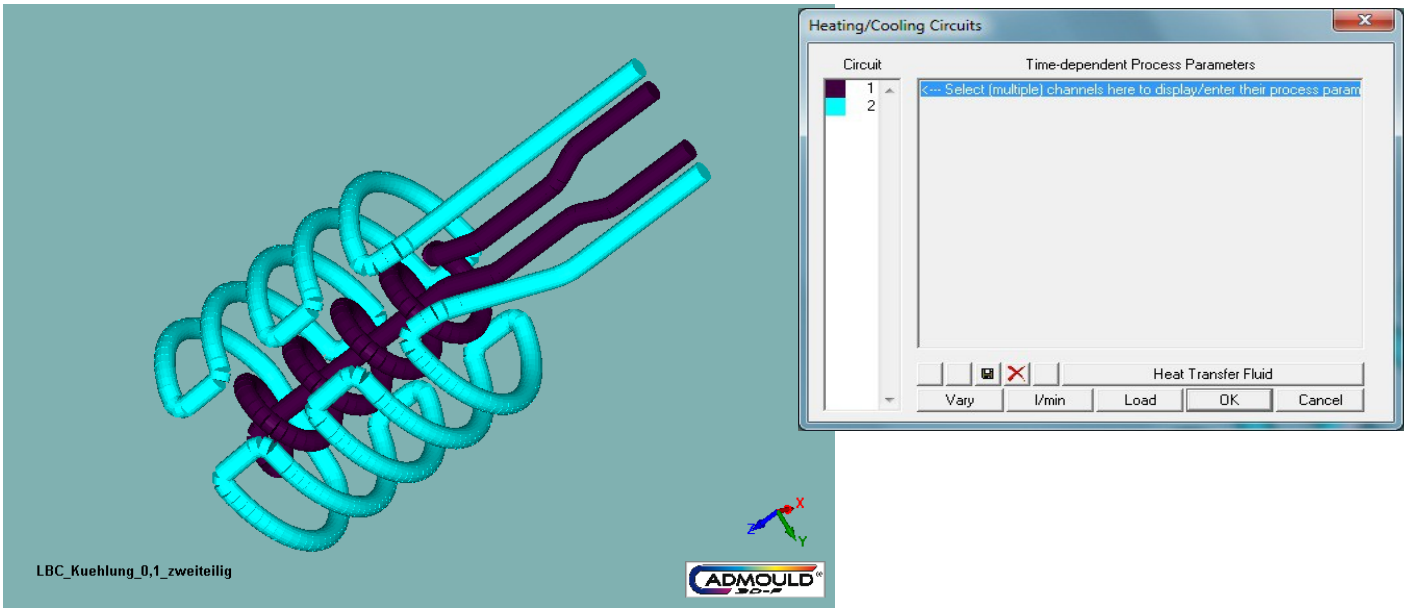
⇒ After the conversion you get the following display of the converted h/c-system:Now clear the table with the button with the red cross.

6. Repeat the steps 3. - 5. for all h/c-circuits of your h/c-system.
7. After you have converted all circuits of your system, you only have to delete the STL file. For this open the function “Delete” from the icon bar “Edit” and delete the loaded STL files.

Remark: H/C-Systems loaded as STL file appear in the menu delete as “Parts” and not as “H/C-Segments”. So you have to delete “Parts”.

8. Open the function “H/C Circuits” from the button bar “Wall Temperature” to check the system for failures.

⇒ You get the following display:

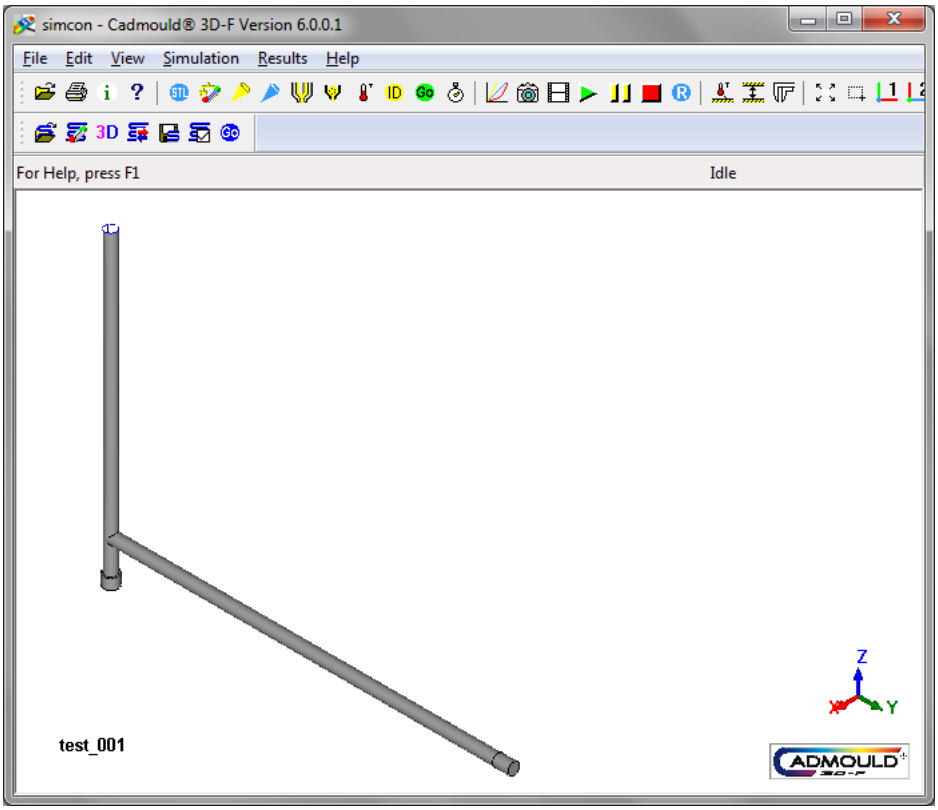


If there are no failures displayed in the window, the conversion was done correctly.

7.8.2 Drilled cooling channel

1. Click the button **<File – Import – STL >** and select the STL-File, which contains your cooling system
2. The window to scale up your part appears. Unless otherwise requested, leave the setting to mm and click **<Ok>**

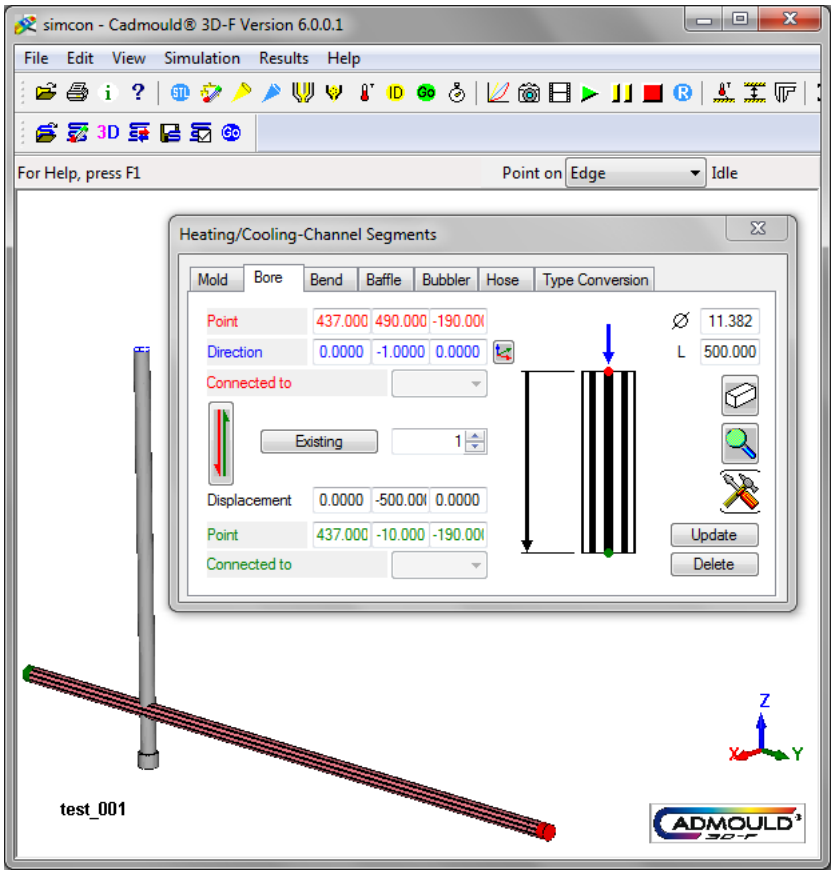
Now, you have your cooling system in your Working Area.



3. Click the button **<Edit – Geometry – Heating/Cooling Channel – Segments>** and select in the Status Bar <Point on **Edge**>
4. Click while holding the Ctrl-key on the edge-loop center of your cooling channel.

Immediately the hole center (center mark on the mould surface), direction and diameter of the hole appear in the dialog.

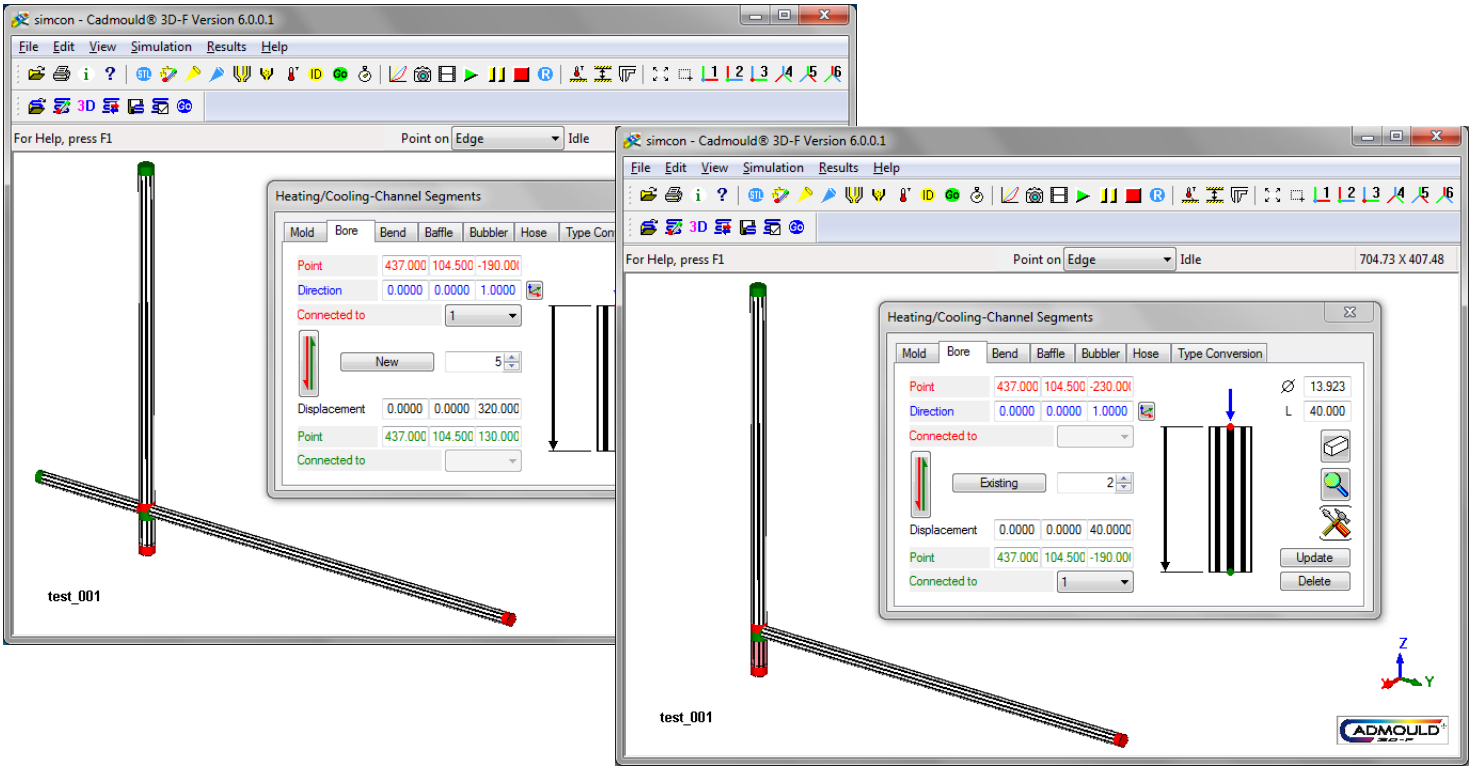
5. Enter a very high value for the length an click **<Create>**. The protruding ends will be cut later.



6. Do the same with the transverse bore. The previous segment gets trimmed automatically.

7. Select the protruding end you want to delete by click on the segment while holding the Ctrl-Key. Then click the button “Delete”.

8. Finally you have to delete the imported STL. Therefore go to **<Edit – Geometry – Tools – Delete>** select the STL-Part and click on “Delete”.

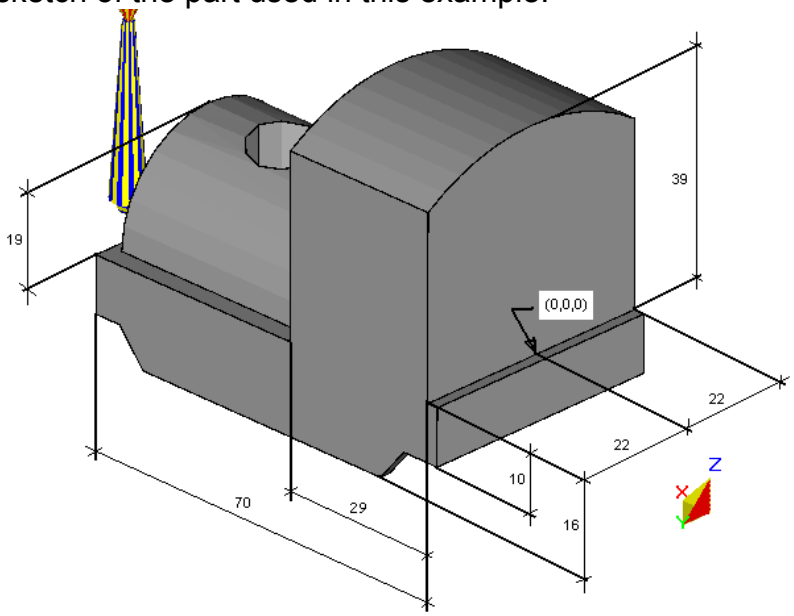


7.9 Example of a heating/cooling system

During the construction of a heating/cooling system you should be sure about the dimension of your part and the coordinate system origin you defined during the construction of your part at the CAD-system.

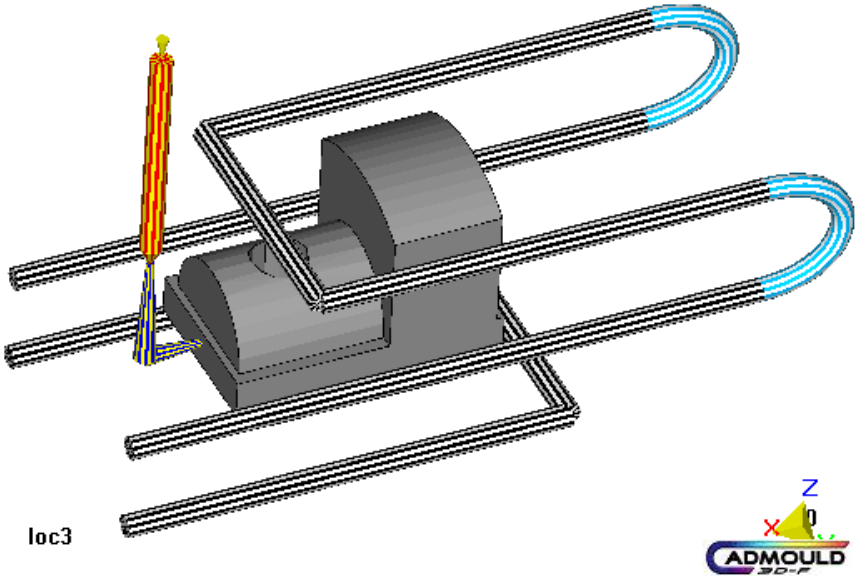
For a new definition of the coordinate system origin consider please chapter "Transform".

A sketch of the part used in this example:



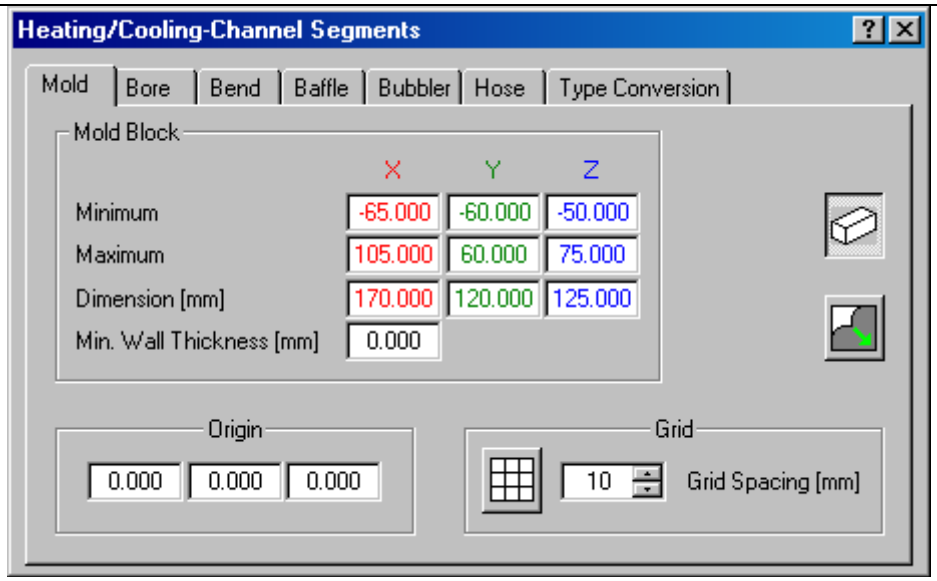
This is the goal.

Example: Loc3 with a runner system and a H/C system



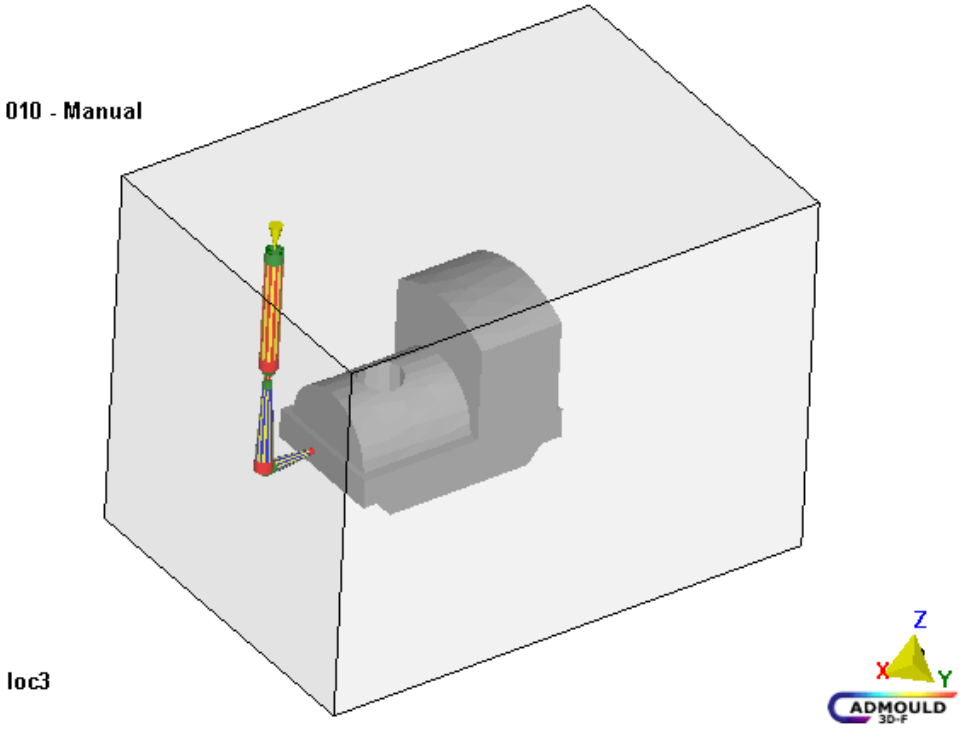
1) Click "Heating/Cooling System".

   ⇒ The dialog "Heating/Cooling-Channel Segments" appears on the screen.



The dimensions of the mold block were in this example already defined. The visibility of the mold block was activated by pushing the button.

The display shows now the following:



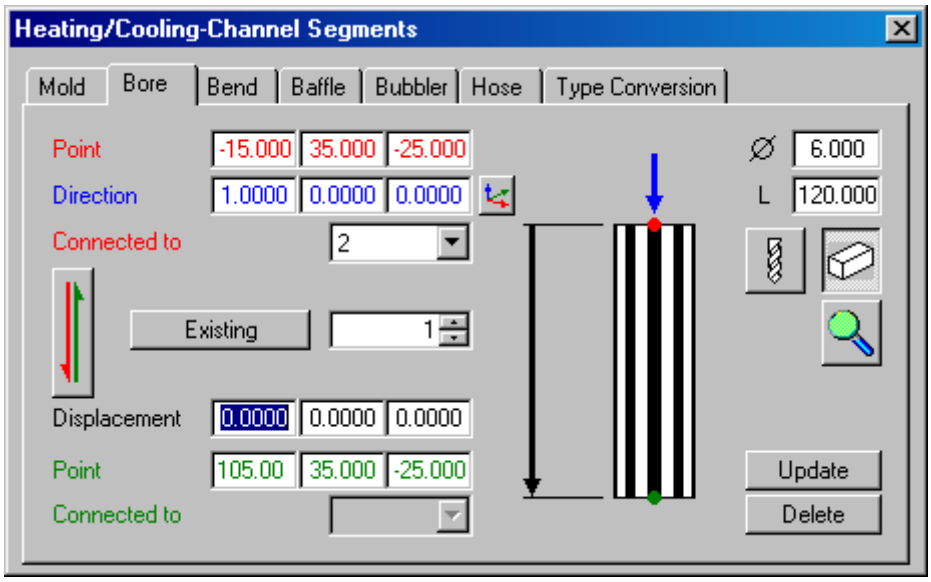
Modelling the first heating/cooling segment:

1) Change to the register "Bore".
2) Activate the "red" point, by putting the cursor into one of the coordinate fields.
3) Enter the coordinates of <Point 1> (red). To get the coordinates of the second point, you can either:

- ♦ enter them directly,
- ♦ enter the direction (e.g. by mouse click selection) and the length,
- ♦ enter the displacement.

4) Enter the diameter of the bore segment.

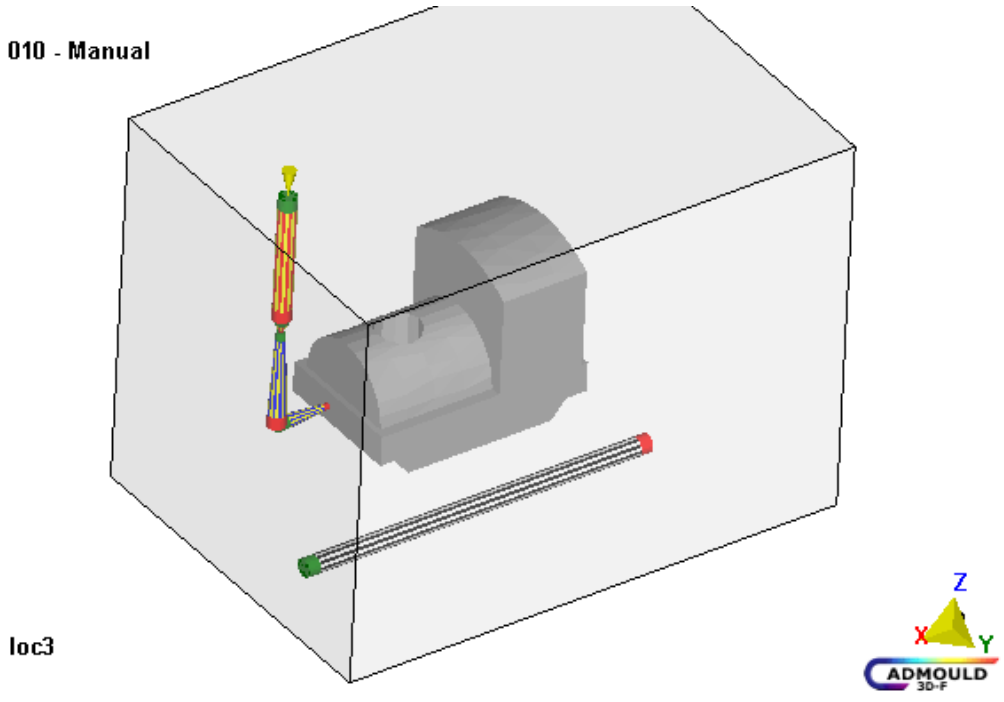
5) All parameters are given, when the <Create> button turns active. (Make sure, that all parameters are correct before you click <Create>.)



6) Click <Create> to get the segment on the screen.

Segments, that already exists (the input field for the segment number indicates "Existing" and a segment is highlighted red.) can be updated or deleted.

⇒ You get the following result on the screen.



Next Segment:

1) Enter the coordinates of <Point 1> (red) by using the "Select Function".

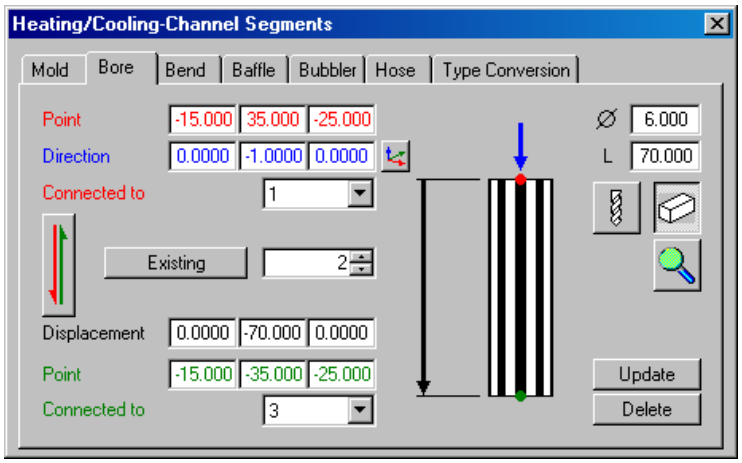
Note: To digitize points on the part, the mold block must be undisplayed.

(*See* chapter: Select Function)

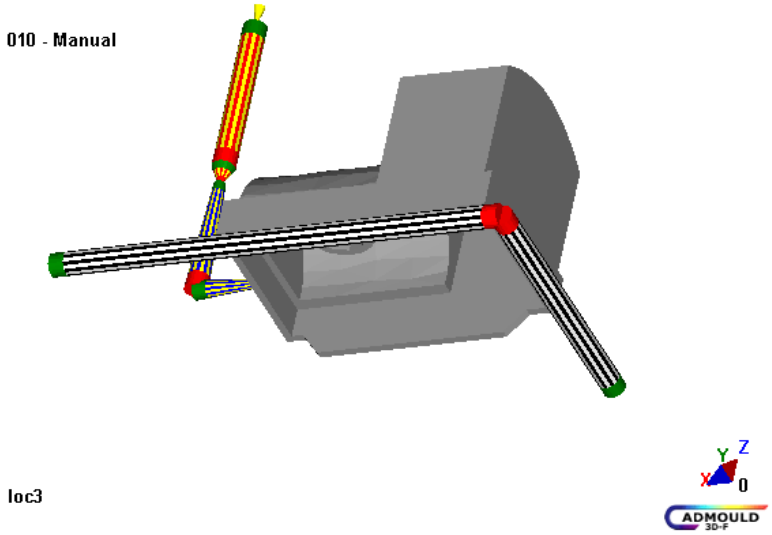
2) Get the coordinates of the second point, you can either:

- ♦ enter them directly,
- ♦ enter the direction (e.g. by mouse click selection) and the length,
- ♦ enter the displacement.

3) Enter the diameter of the bore segment.



4) Click <Create>.



Next Segment:

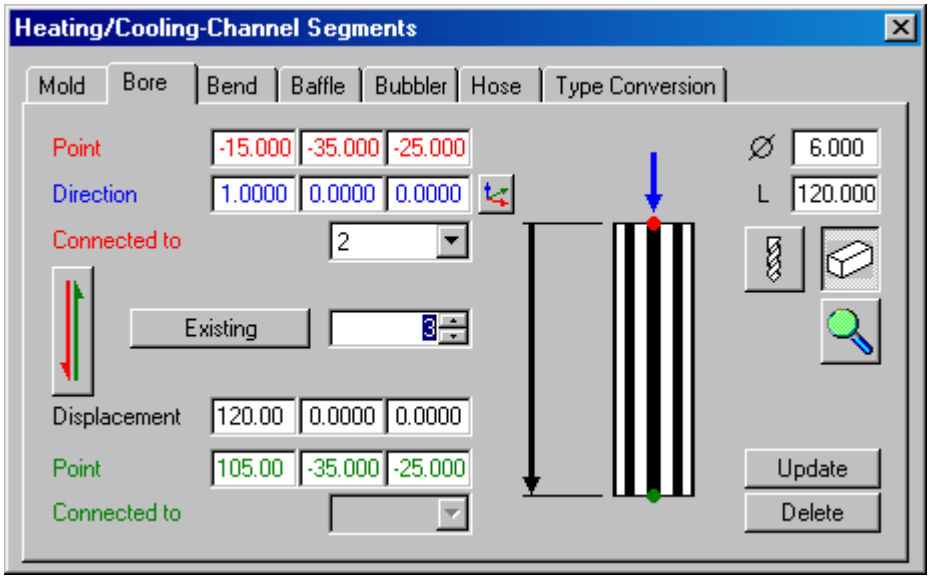
5) Enter the coordinates of <Point 1> (red) by using the “Select Function”.

(*See* chapter: Select Function)

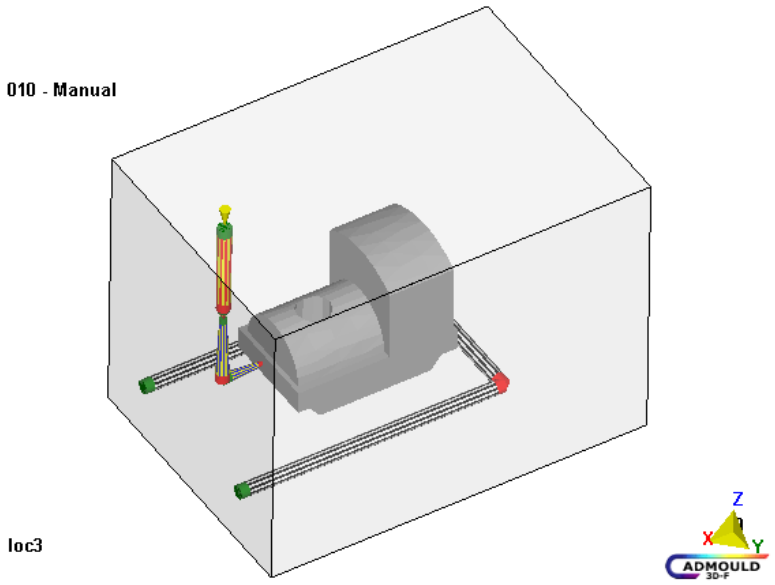
6) Get the coordinates of the second point, you can either:

- ♦ enter them directly,
- ♦ enter the direction (e.g. by mouse click selection) and the length,
- ♦ enter the displacement.

7) Enter the diameter of the bore segment.



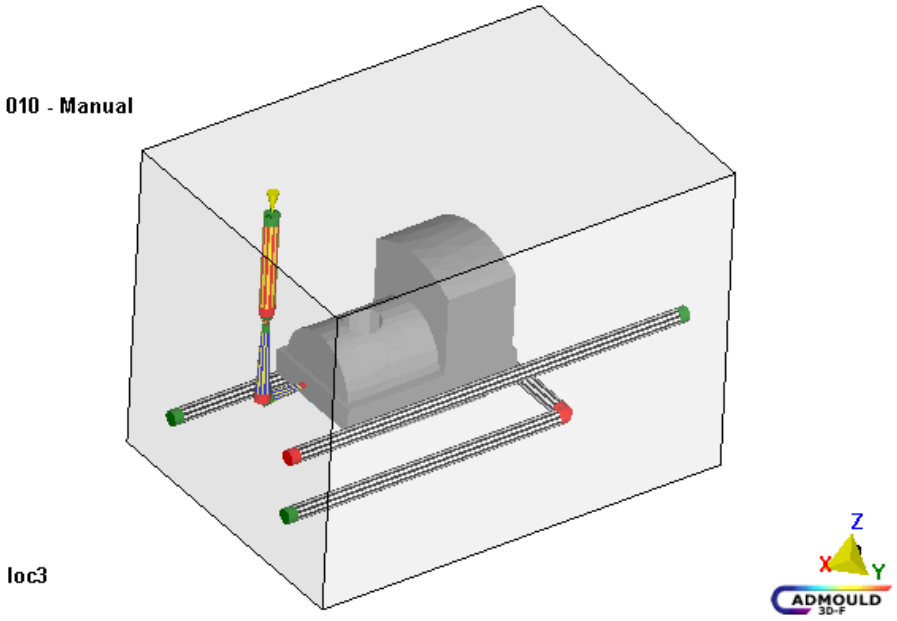
8) Click <Create>.
9) Reactivate the display of the mold block.



Next Heating/Cooling Circuit:

The next circuit will be designed with the new design aid functions for heating/cooling circuits. Therefore will first be drilled two bores through the part in X-direction.

1) Activate the function "Drilling" on the register "Bore".
2) Digitize the point on the mold surface where the drill should start.
   (In this example: Y≈35; Z≈0)
3) All further input data will be filled in automatically, as you see in the next picture.
4) Click "Create".

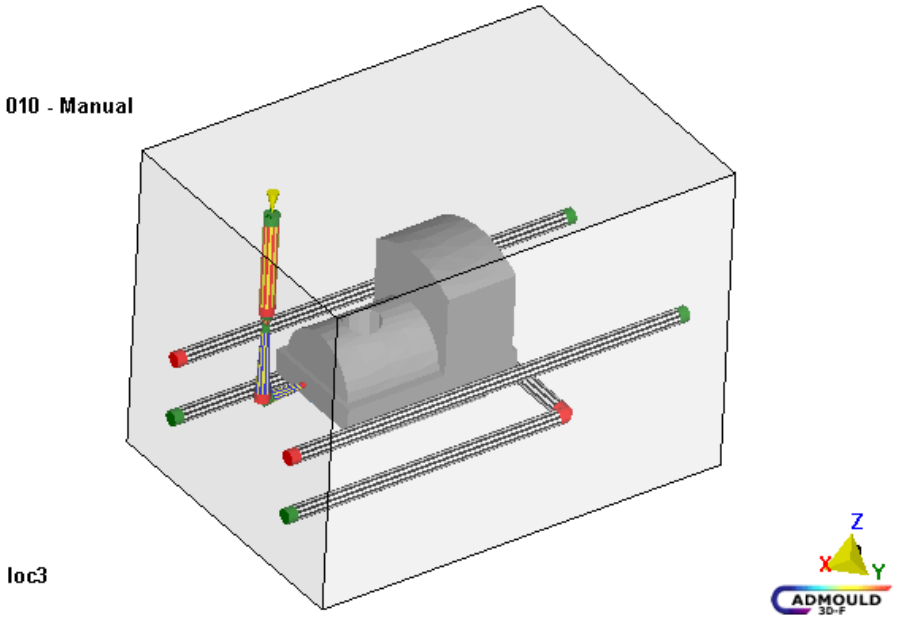


5) Repeat this no for the opposite of the part.

6) Digitize the point (Y≈-35; Z≈0) on the mold surface.

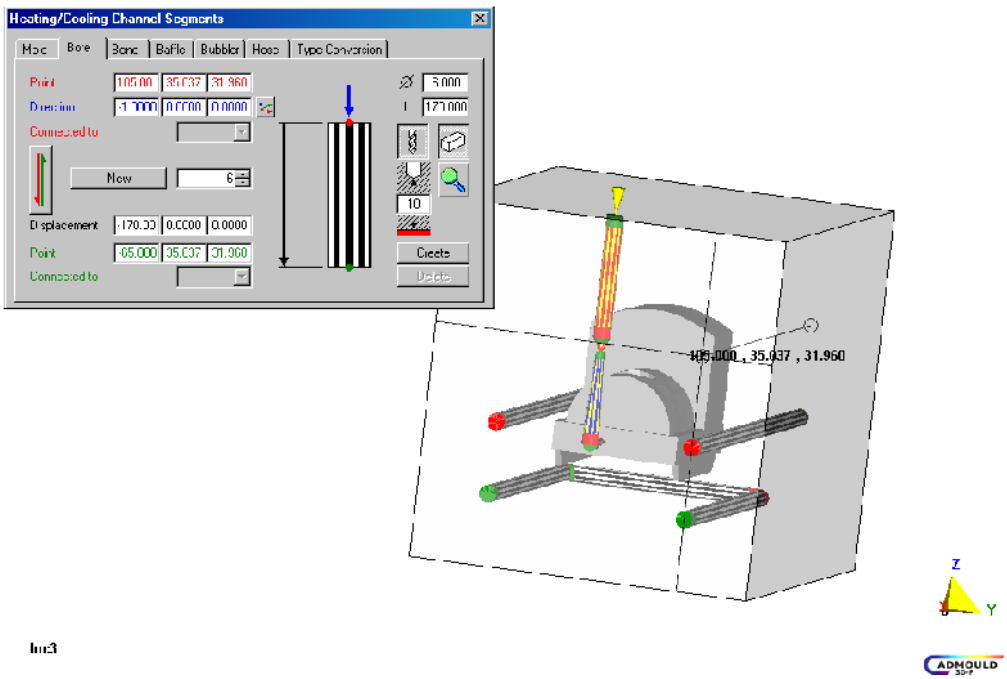
Remark: The cursor must be in one of the coordinate fields of the "red" point to create a dril.

7) Click "Create".



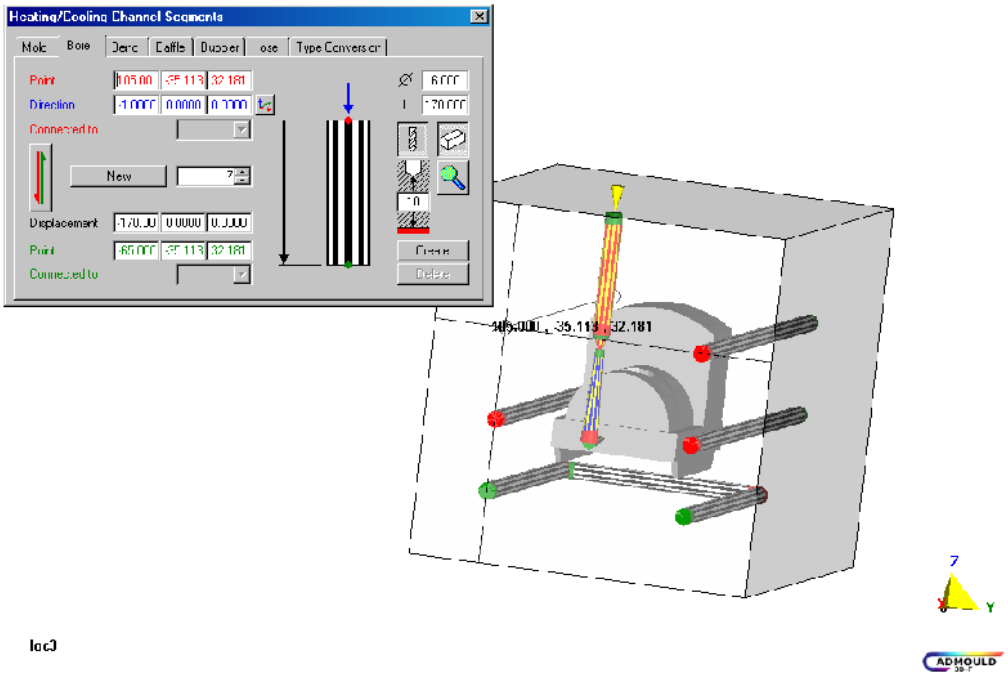
Next Segments:

1) Digitize the first point of the drill on the mold surface. (Y≈35; Z≈32)



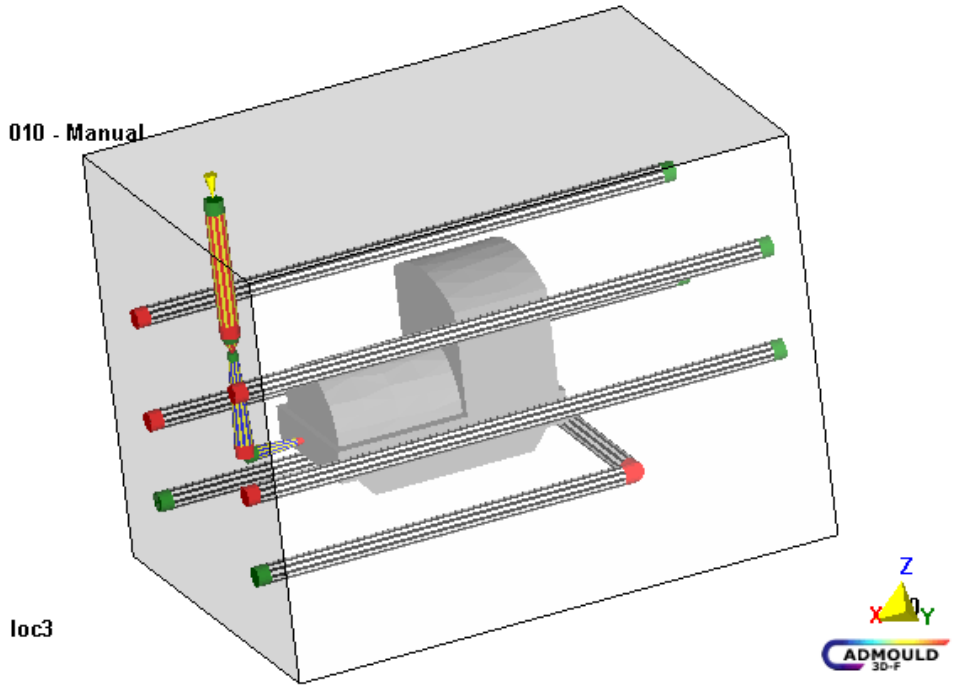
2) Click "Create".

3) Digitize the first point of the next drill on the mold. (Y≈-35; Z≈32)



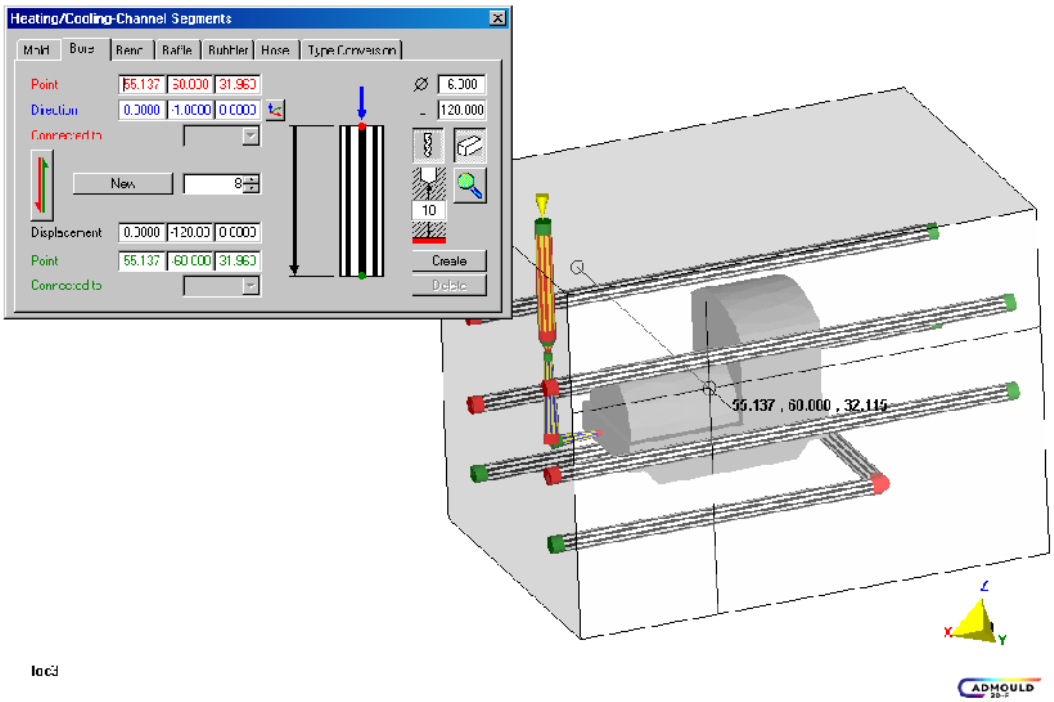
4) Click "Create".

The heating/cooling system is now shown like this:



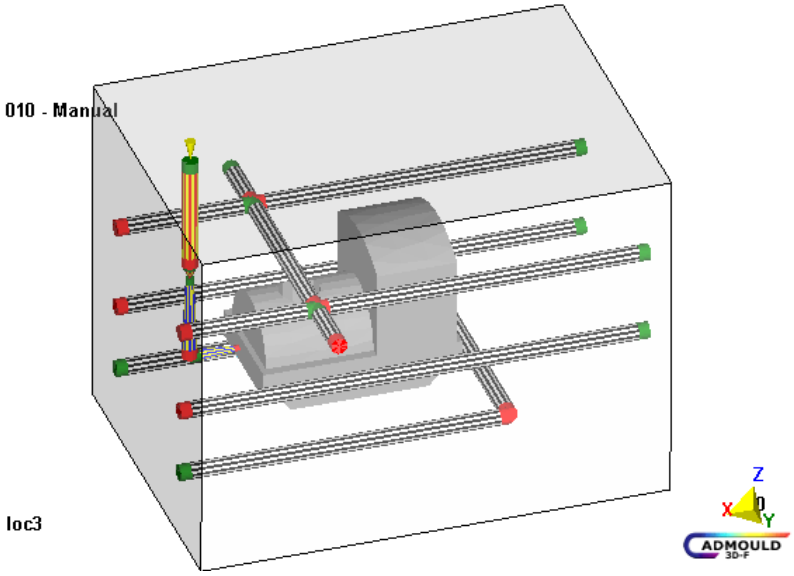
Connection drill between the both upper segments:

5) Digitize the starting point of the drill on the side of the mold.
(X≈55; Z≈32)



6) Click "Create".

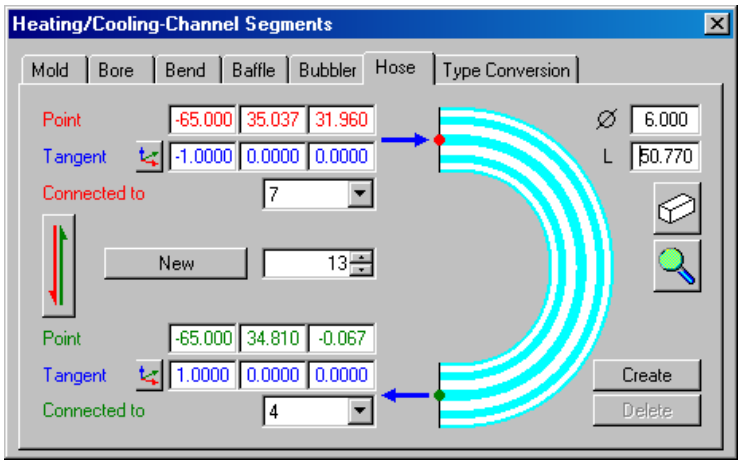
You have now the following result:



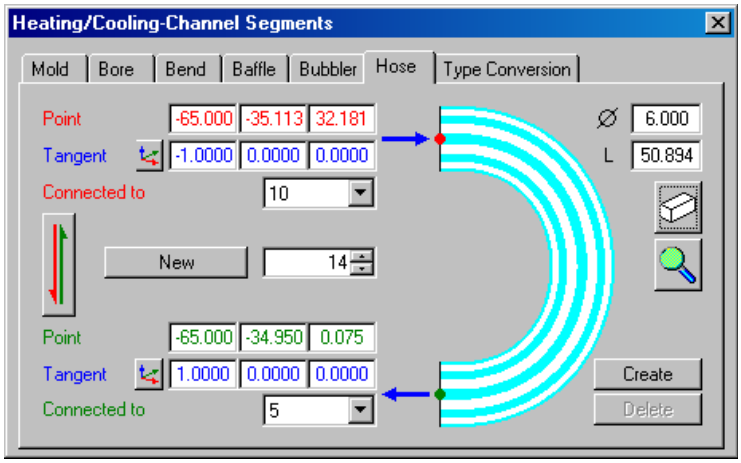
The crossing drills were cut with each other and you can delete the superfluously segments afterwards. Before we do this, we will connect the upper circuits with hoses.

1) Jump to the register "Hose".
2) Deactivate the display of the mold block.
3) Enter the coordinates of <Point 1> (red) and <Point 2> (green) by using the "Select Function" (Ctrl + left mouse button). Enter the diameter.
4) Leave the last field with the "Tab" Button.

Note: The direction of the tangents and the length of the hose will be calculated automatically by CADMOULD® 3D-F.

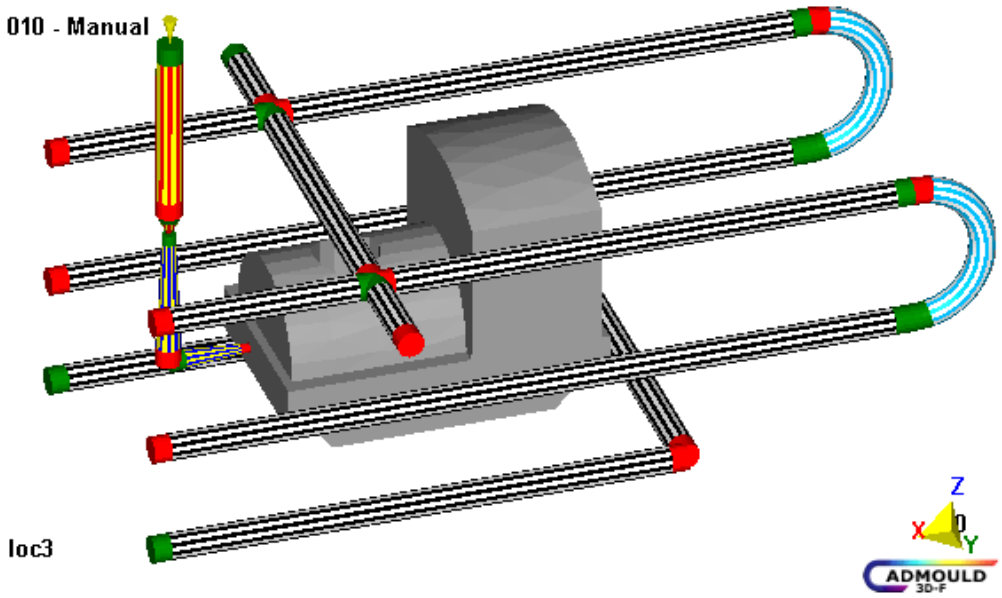


5) Click "Create".
6) Repeat the points 3) - 4) for the second hose.



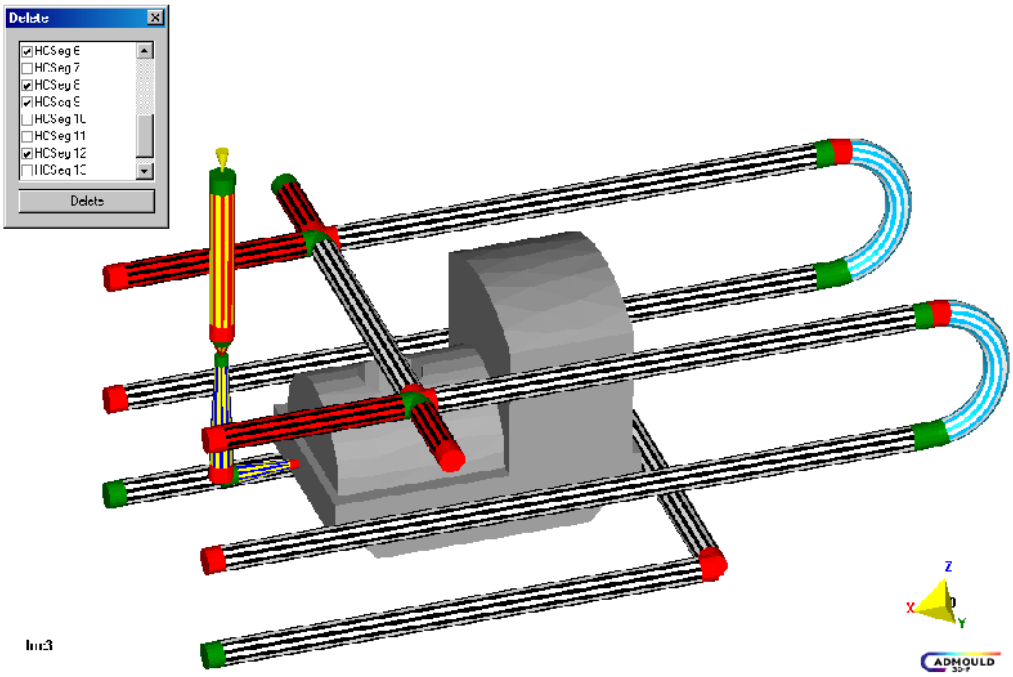
7) Click "Create".

The result looks now like this:

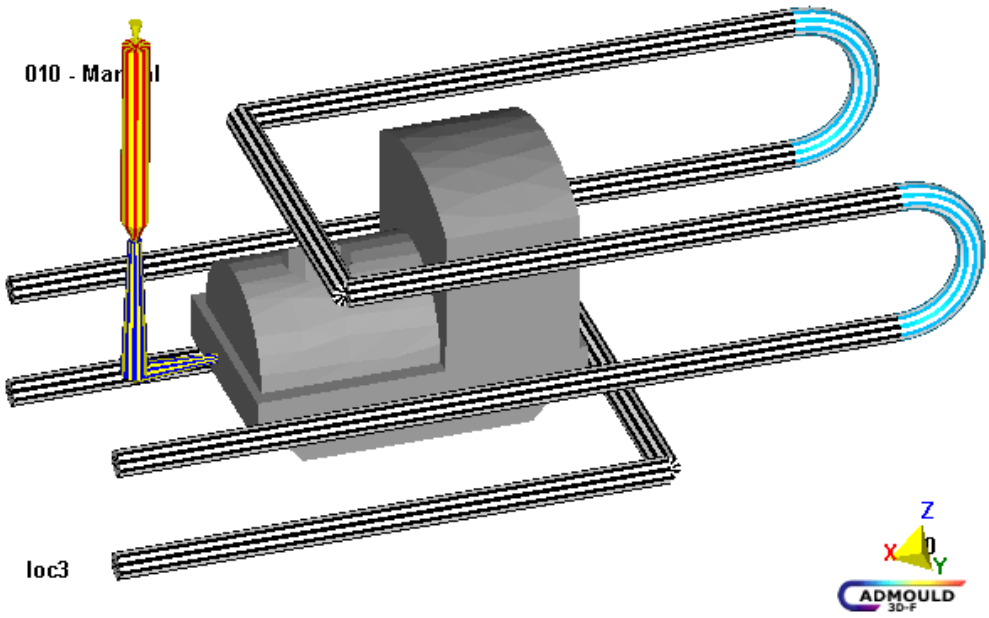


Deleting the superfluous segments:

1) Open the function "Delete" by pushing the button in the button bar.
2) Select the segments which should be deleted with "Ctrl + left mouse button".



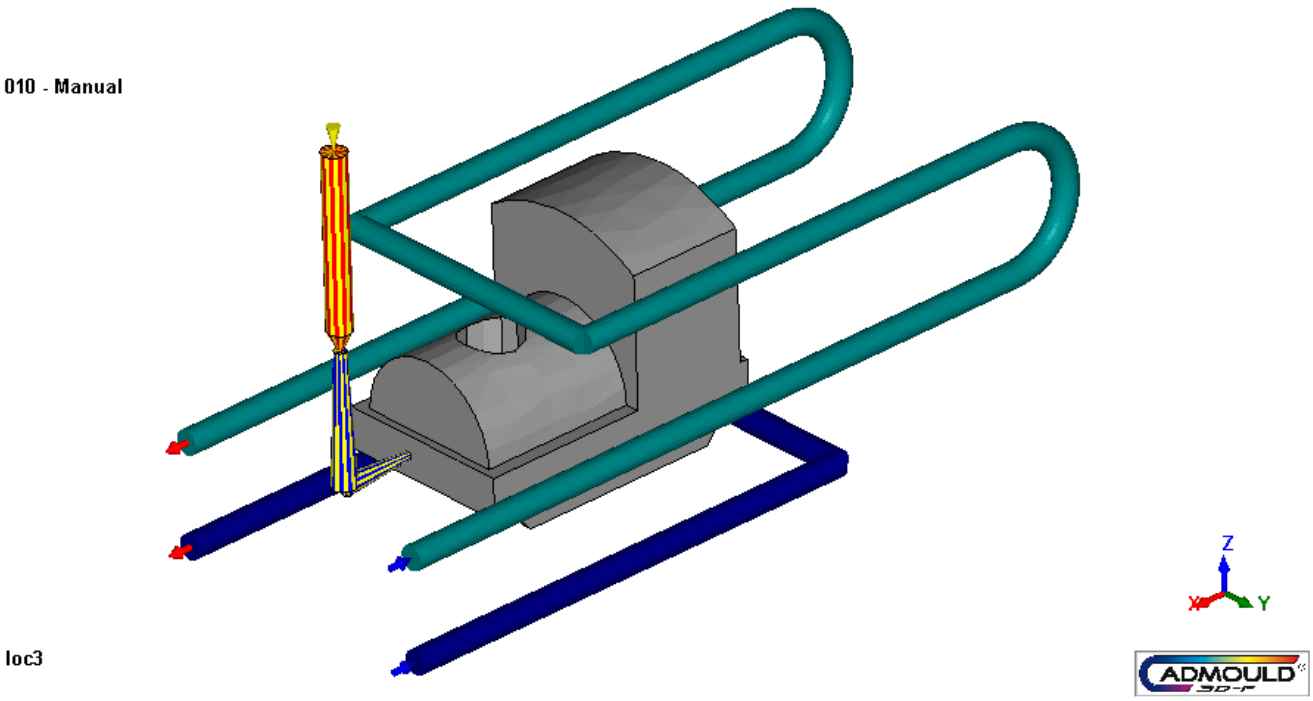
3) Click "Delete".

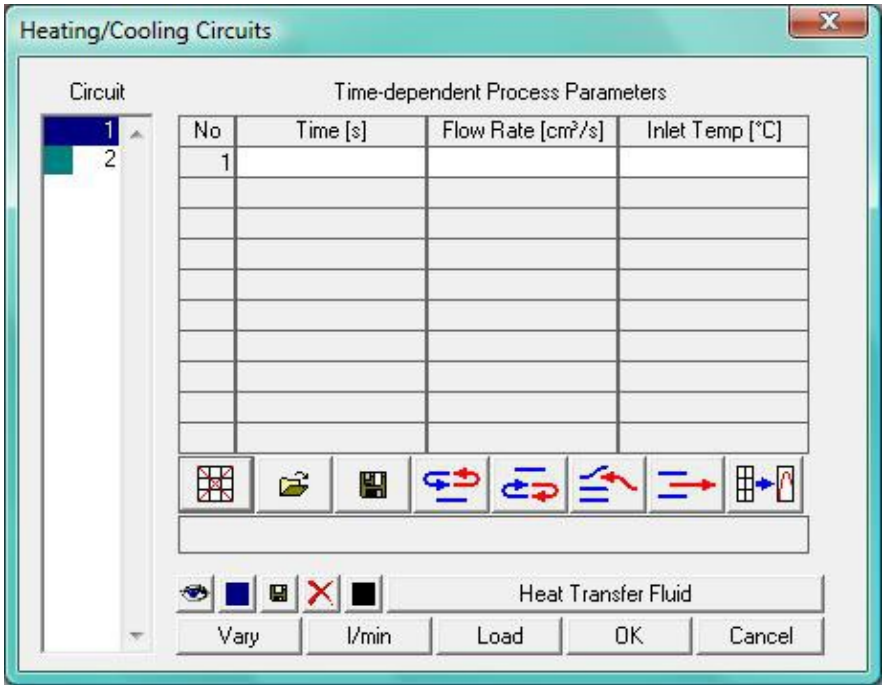


The goal is reached. CADMOULD® 3D-F recognizes automatically all existing heating/cooling circuits and numbers them.

4) Click on “Heating/Cooling Circuits” to check the circuits and to enter the parameters.

(*Please see also* chapter: Heating/Cooling Circuits)





8. Stack and compute projects

(Separate auxiliary module of CADMOULD® 3D-F)

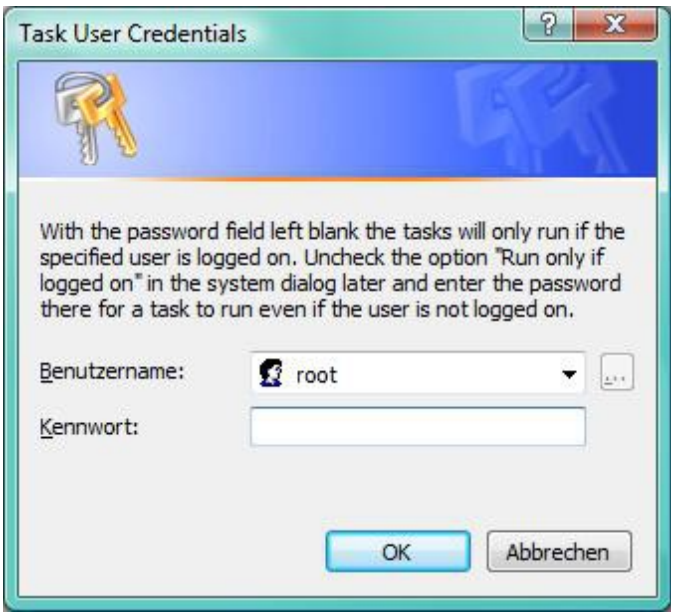
The "CADMOULD® 3D-F Task Schedule" is a stand-alone program, which can be started over the "Start"-menu of windows or in CADMOULD® 3D-F by pushing the button "CADMOULD® 3D-F Task Schedule". Several projects can be stacked and be computed successively - also during your absence.
Use for it the CADMOULD® 3D-F **Task Schedule**.

Note: The "CADMOULD® 3D-F Task Schedule" can be started with user-specific settings. Please see chapter "Command Line Options"

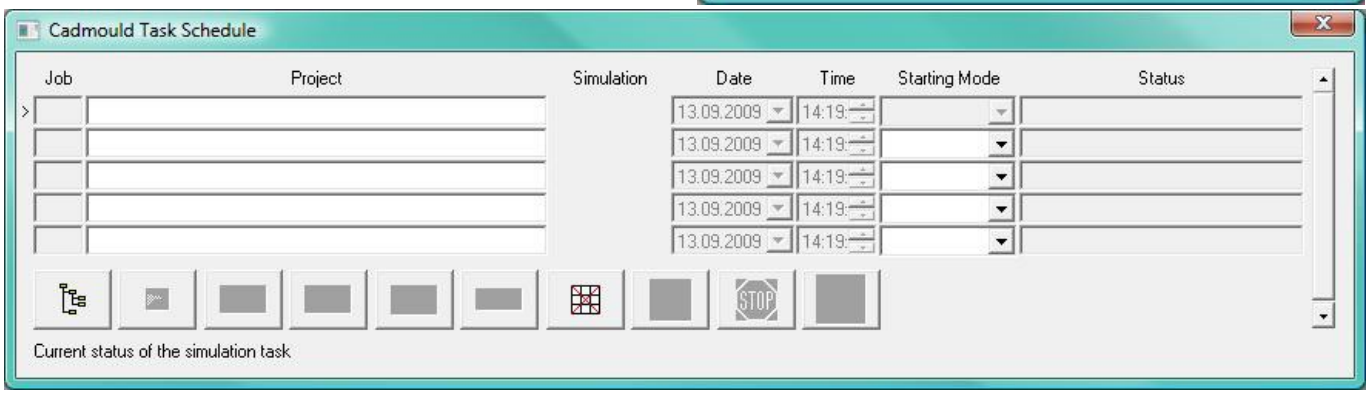
(1) Start the "CADMOULD® 3D-F **Task Schedule**".

⇒ The window "**Task User Credentials**" appears on the screen.

If your scheduled simulations should be calculated while the registered user is not logged in, you have to enter the password.
If not, you can continue without a password.


Task User Credentials
With the password field left blank the tasks will only run if the specified user is logged on. Uncheck the option "Run only if logged on" in the system dialog later and enter the password there for a task to run even if the user is not logged on.
Benutzername:
root
Kennwort:
OK
Abbrechen

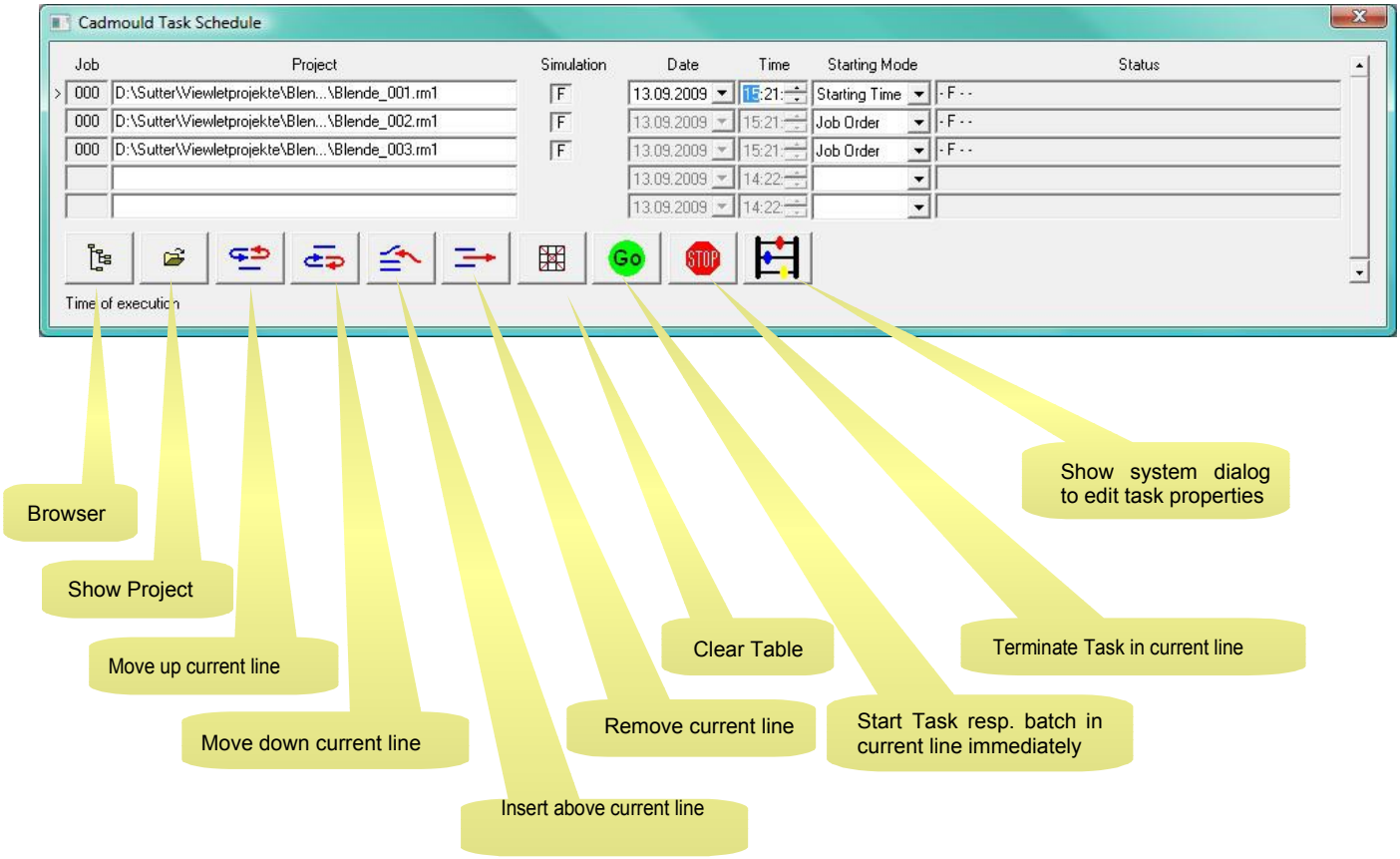
⇒ The window "**Cadmould Task Schedule**" appears on the screen.



The CADMOULD® 3D-F **Task Schedule** always shows all CADMOULD® 3D-F jobs present in the memory ("task" directory) and their status.

If some CADMOULD® 3D-F jobs had been stored, then these last in the CADMOULD® 3D-F **Task Schedule** listed as follows.

Example:



Browser
Show Project
Move up current line
Move down current line
Insert above current line
Remove current line
Clear Table
Start Task resp. batch in current line immediately
Terminate Task in current line
Show system dialog to edit task properties

Column:	Description:
Job:	internal number; Several tasks (variant calculations) can be combined into one job, if they are to be computed successively (starting status: Sequence).
Project:	Projects with full path information. (*See* chapter: Open)
Simulation:	CADMOULD® 3D-F process simulation modules (can be switched on and off by mouse-click). Dependent on your defaults of the process parameters and your license process simulation modules stand for you to arrangement. **T = Wall Temperature Analysis** (*See* chapter: Heating / Cooling - Circuits; Options – Wall Temperature Analysis) **F = Cavity Filling Analysis** **P = Packing** **W = Shrinkage and Warpage Analysis** The process parameters of fill -, packing -, shrinkage and warpage analysis, have to be stored already assigning a project number. (*See* chapter: Process; Description)
Date:	The date, at which the simulation is to be started.
Time:	The time, at which the simulation is to be started.
Starting mode:	Starting time = beginning of simulation; new order knowing number (job) Sequence = the projects are processed in sequence.
Status:	Current conditions of the tasks (of the project).

(2) Click on <Browser>.

⇒ The window "Open" appears on the screen.

(3) Mark a project and click on <Open>.

⇒ The project appears in CADMOULD® 3D-F **Task Schedule**.

(4) Position the Courser (pointer of mouse) in the next free line and repeat the steps 2, 3, 4, to all projects you would like to compute, until all of them stand in the CADMOULD® 3D-F **Task Schedule**.

(5) Give the starting date, the starting time and the starting mode,

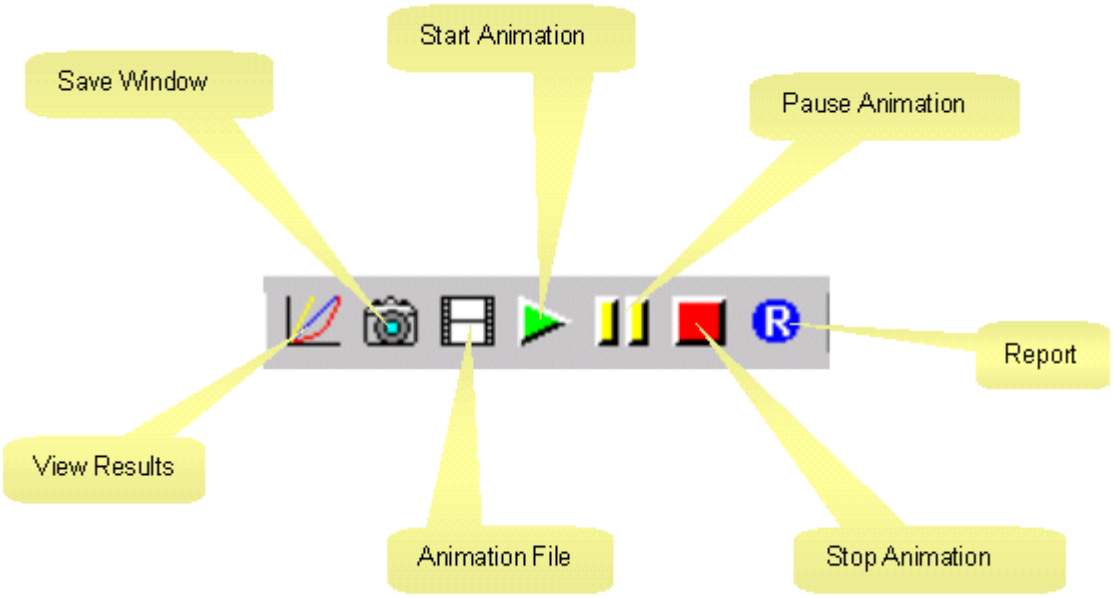
or

you start the job in the current line immediately by clicking on <GO>

"Start Task resp. batch in current line immediately".

9. Result selection

You can view and document results in several ways.



Click "View Results".

⇒ The window "Result Selection" appears on the screen.

Different *Result Groups* are available. In each group there are several *Result Details*.

Visible results can be varied and additional information displayed using the controls below these selection fields.

9.1 Result groups

Mind that depending on licensing and options selected for simulation the offered choice of result groups may vary!

- Result Summary
- Cavity
- Flow Front
- Snapshot Filling
- Filling Temperature (interior)
- Filling Velocity (interior)
- Filling Orientation Degree
- Diagrams
- Residual Stress
- Miscellaneous
- Part Quality
- Snapshot Packing
- Packing Temperature (interior)
- Packing Velocity (interior)
- Packing Orientation Degree
- Shrinkage and Warpage - Cooling
- Shrinkage and Warpage (Ejection)
- Shrinkage and Warpage (Final)
- Residual Stress in Layers –5 to +5
- Wall Temperature Analysis
- Heating/Cooling System
- Sensor

Result files can be converted into text files.
(*See* chapter: Convert / CAR ->Txt)

9.2 Result details

Mind that depending on licensing and material properties available the offered choice of result details may vary.

Cavity:	Part without results
Flow Front:	Time When Filled [s] Level When Filled [%] Filled by Gate Temperature When Filled [°C] Pressure Loss [bar] Shear Stress When Filled [kPa] Velocity When Filled [mm/s]
Snapshot Filling:	Temperature [°C] Pressure [bar] Shear Stress [kPa] Velocity [mm/s] Frozen Layer Thickness [%] Free Cross Section [mm] Volume Shrinkage [%]
Filling Orientation Degree:	Averaged
Miscellaneous	Max. Temperature [°C] Max. Shear Rate [1/s]
Diagrams:	Injection Pressure [bar] Flow Rate [cm³/s] Injection Power [kW] Clamping Force X [kN] Clamping Force Y [kN] Clamping Force Z [kN] Level of filling [%] Mass [g] Mass - Gate 1 [g] Mass Flow - Gate [g/s] Force Xn on object group [kN] Force Yn on object group [kN] Force Zn on object group [kN] Moment Xn on object group [Nm]

	Moment Yn on object group [Nm]
	Moment Zn on object group [Nm]
Residual Stress (element and node based):	Avg. Hydrostatic
	Max. Von Mises
Part Quality:	Filling Problems
	Sinkmarks
	Ejection Time [s]
	Sealing Time [s]
	Freezing Time [s]
	Gradient Volume Shrinkage [1]
Snapshot Packing:	(*See the Snapshot Filling results.*)
Packing Orientation Degree:	Averaged
Shrinkage & Warpage (Cooling):	Max. Temperature
	Min. Temperature
Shrinkage and Warpage (Ejection and Final):	X-Deformation [mm]
	Y-Deformation [mm]
	Z-Deformation [mm]
	Deformation [mm]
	Shrinkage [%]
	X-Warpage [mm]
	Y-Warpage [mm]
	Z-Warpage [mm]
	Warpage [mm]
	Curvature Change [%]

Filling Temperature (interior); Filling Velocity (interior)
Packing Temperature (interior); Packing Velocity (interior)

Residual Stress in Layers –5 to +5 without 0:	Avg. Hydrostatic
(Element based and node based)	Max. Von Mises
Wall Temp. Analysis Diagram:	Max. Temperature [°C]
	Min. Temperature [°C]

Heating/Cooling System:	Flow Rate [cm³/s]
	Reynolds number [1]
	Heat transfer coefficient [W/(m²K)]
	Heat flow efficiency [1]
	Temperature [°C]
Sensor:	Pressure [bar]
	Temperature [°C]
	Viscosity [Pa·s]
	Shear Rate [1/s]
	Specific Volume [cm³/g]

(*See* chapter: Description of results)

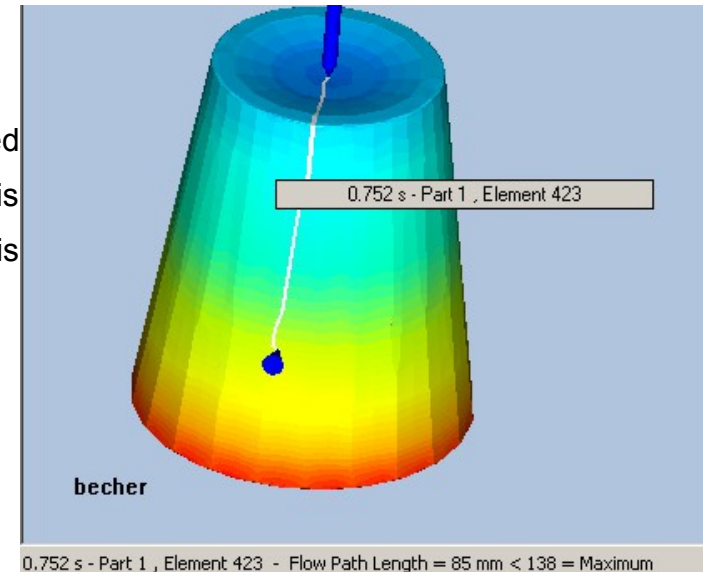
Note: Hitting the button “Detail”, information pops up. Eg: "Pressure [bar]" is the new default result detail of result group "Snapshot Filling."
The standard result detail is a user preference and will be saved in the registry. If there is no entry, the first result column of the result group will be shown.

9.2.1 Local Results

You can display numerical results at locations of interest.
(1) Select the desired type of result in the window "Result Selection"
(2) Keep the <Ctrl> key pressed.
(3) Click the location of interest on the part and keep both keys pressed.
⇒ A blue cone appears. See the following example.

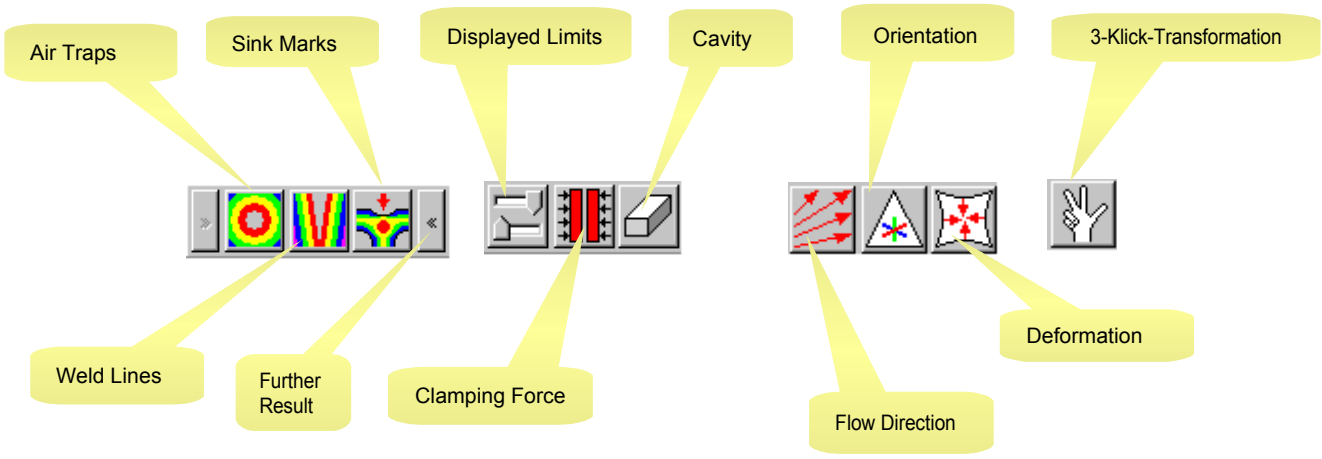
The result at the desired location is displayed.

(*Example:* The flow front reaches the picked location after 0,752s. The flow path length is 85mm and the max. flow path length is 138mm.)



9.2.2 Result Display

In the window for result selection there are scrollable buttons allowing to add or to modify the selected result.



9.2.2.1 Air Traps

Click the button "Air Traps" in the window: "Result Selection".

The places, in which air traps may occur, are marked by small cones on the part.

The color of the cones can be modified by the user. The default is white.

(*See* chapter: "Colors")

9.2.2.2 Weld Lines

Click the button "Weld Lines" in the window: "Result Selection".

The places, in which weld lines develop, are indicated on the part. Depending on the data quality there may be places with "false alarms". It is up to the user to qualify this result.

The color of the lines can be modified by the user.

(*See* chapter: "Colors")

9.2.2.3 Sink Marks

Click the button "Sink Marks" in the window: "Result Selection".

The sink marks on the part are displayed geometrically.

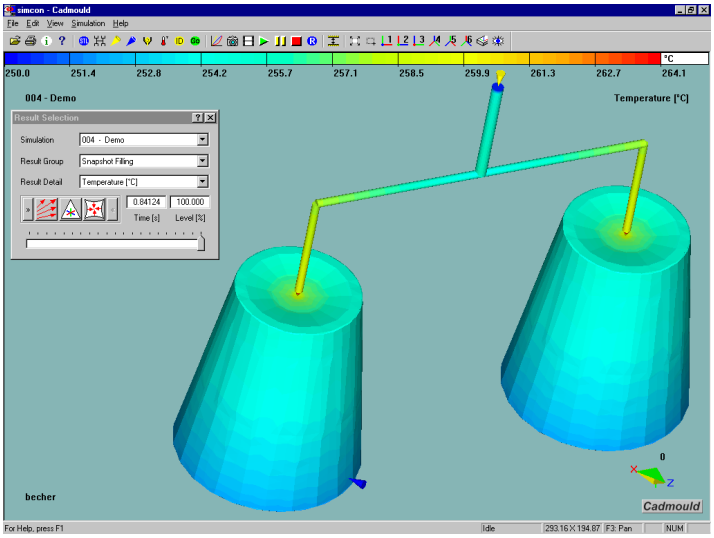
9.2.2.4 Displayed Limits

You can limit the range of the results by the function: "Displayed Limits". The limitation can be accomplished with all results of the categories Flow Front, Snapshot, Miscellaneous and Shrinkage & Warpage.

Here is one example of a limitation of the result: "Snapshot Filling Temperature [°C]"

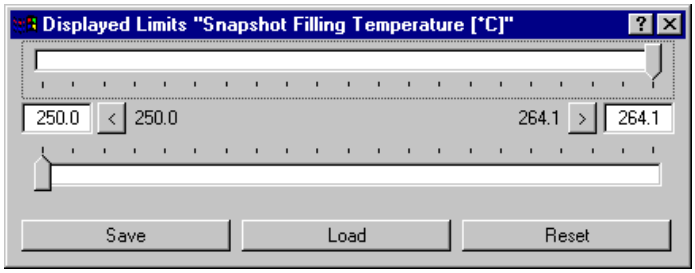
1) Chose the result category „Snapshot Filling" and the result „Temperature [°C]".

Each color is assigned a certain temperature.



2) Click “Displayed Limits”.

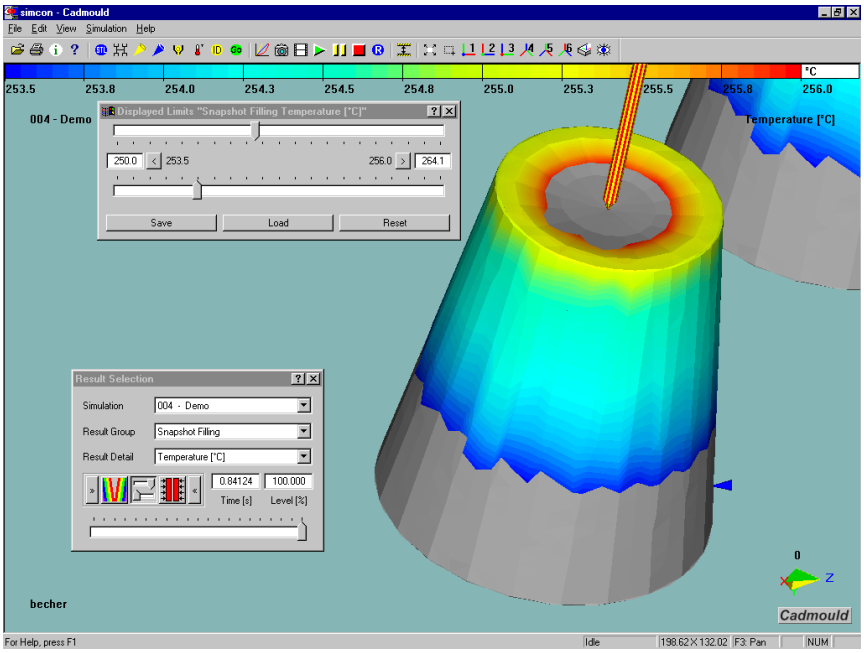
⇒ The window "Displayed Limits “...“" appears on the screen.



In the left and right fields you see the range of temperature occurring during the filling process (250°C to 263.1°C). It is the same range of temperature that is shown in the large color scale below the Menu Bar.

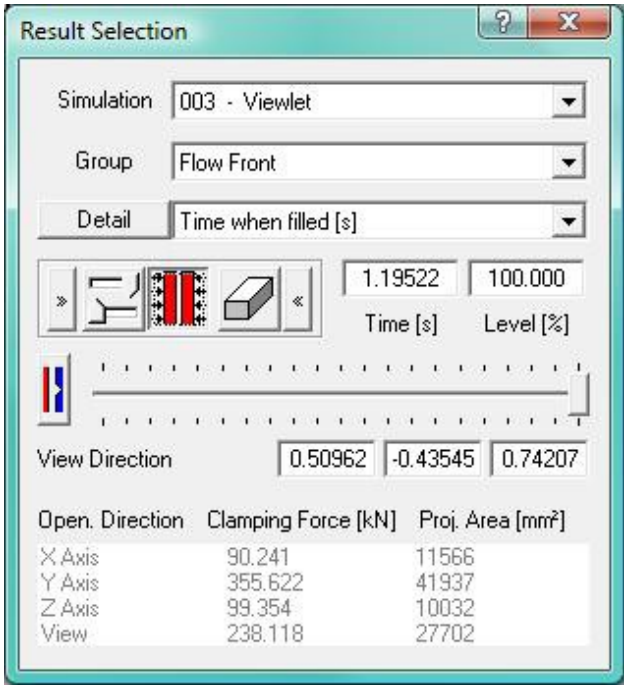
Now, you can limit the results in a range you want to view exactly by comparing the colors, standing for different temperatures, on your part. Use the sliders or the input fields to set the range boundaries. Clicking on “<” or “>” transfers the slider position to the input field allowing a better tuning of the range.

Example: We view the range between 253.5°C and 256°C.



The limited results can be saved and recalled by clicking <Save> and <Recall>, resp.. You cancel the limitation by clicking <Reset>.

9.2.2.5 Clamping Force



Click the button "Clamping Force" in the window: "Result Selection" to get information about the clamping force [kN] and the projected area [mm²] depending on the clamping force direction. Mind that the results do not take account of undercuts.

If the button "Clamping Force" is pushed, the points of load incidence are displayed with red, green and blue arrows.

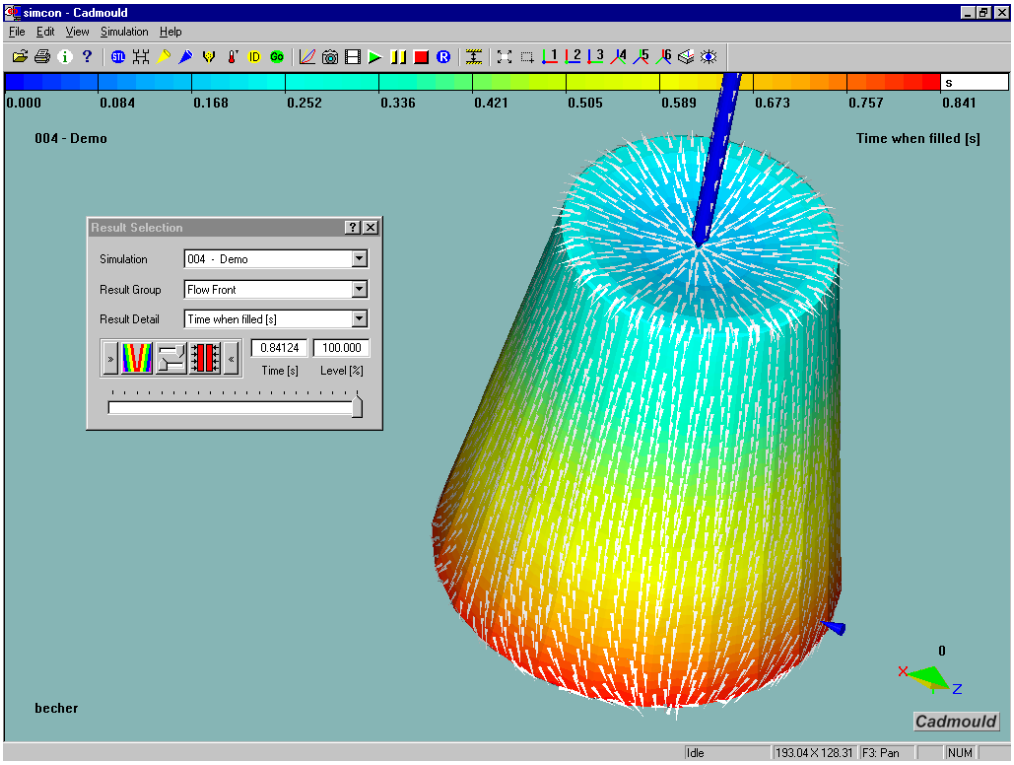
Please regard also chapter: *Clamping Force*

9.2.2.6 Cavity

The button "Cavity" in the result selection dialog serves to display flow directions and orientations without result colors on the part. The arrows and fibers are visible more clear now. It is possible to make an animation with this display option as well.

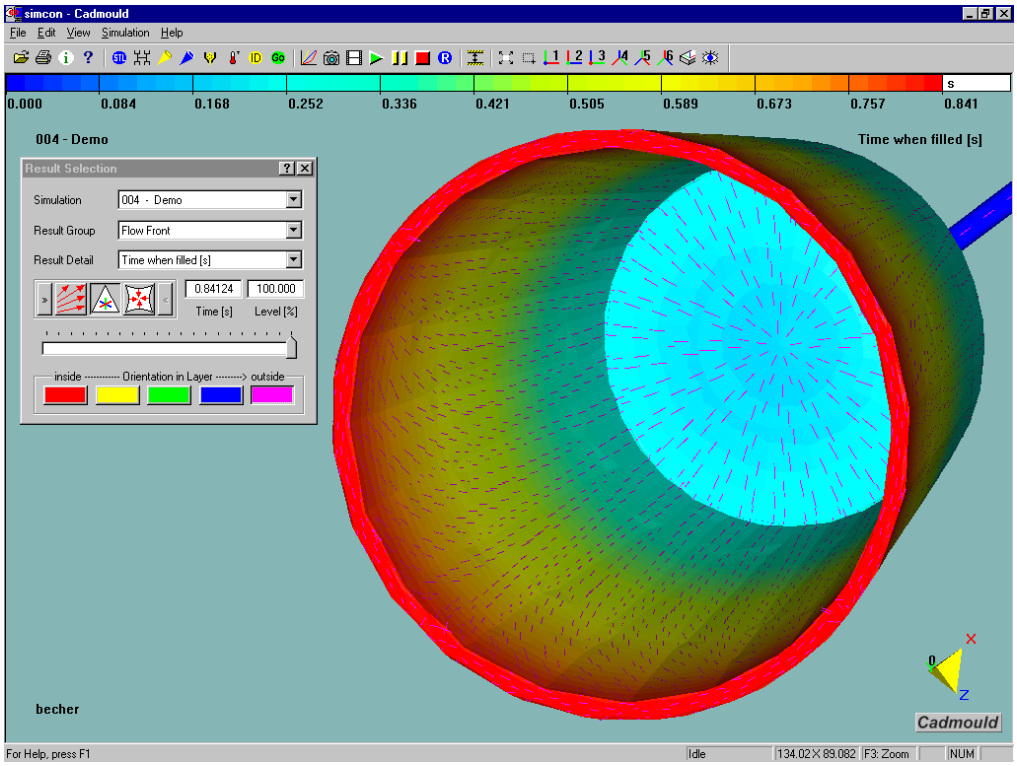
9.2.2.7 Flow Direction

Click the button "Flow Direction" in the window: "Result Selection" to get information about the flow direction in any time of the filling process. The time to view is controlled by the slider.



9.2.2.8 Orientation

Click the button "Orientation" in the window: "Result Selection" and then on one of the buttons (red, yellow, green, blue or magenta) standing for different layers. Equal to the display of directions of flow the orientation may change throughout the process. The time to view is controlled by the slider.

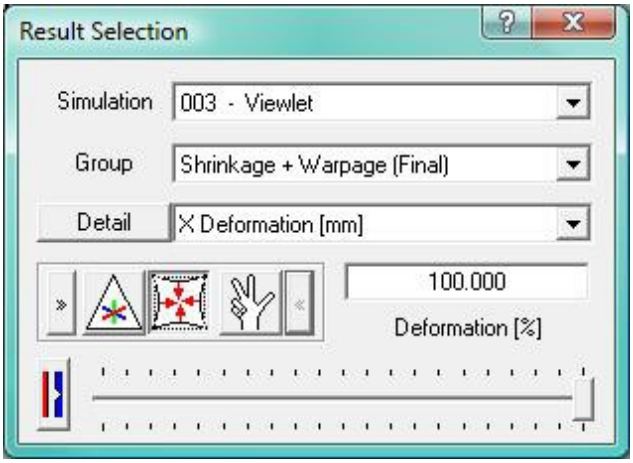


9.2.2.9 Deformation (Shrinkage and Warpage)

If you have shrinking and warpage results available, you have the possibility to examine the deformed shape of your part.

How do you get shrinking and distortion results?

CADMOULD® 3D-F WARP EXPERT user can calculate shrinkage and warpage. Shrinkage and warpage results can be displayed as in the following example described.



Viewing distorted parts

(1) Click „Result Selection".

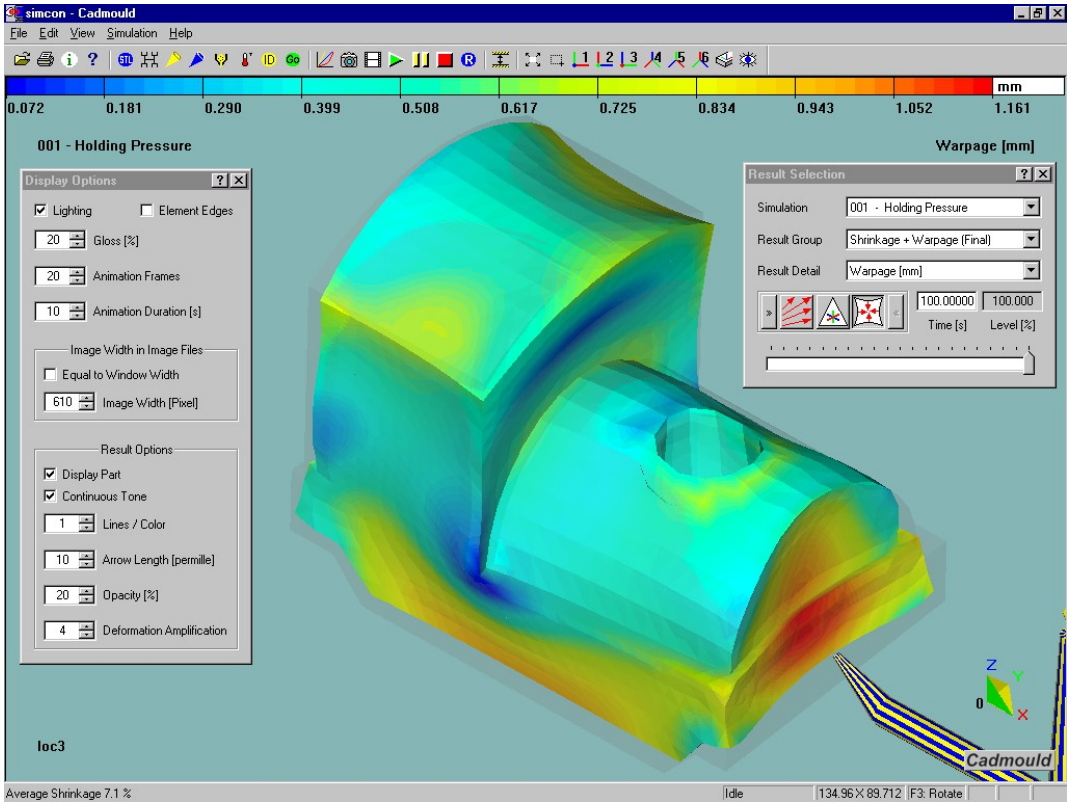
⇒ The window "Result Selection" appears on the screen.

(2) Select the result group "Shrinkage + Warpage (Final)" and a result detail.

(3) Activate the button "Deformation".

(*See* chapter: „Deformation Amplification" and "Opacity [%]")

Example: Shrinkage and Warpage



9.2.2.10 3-Click Transformation

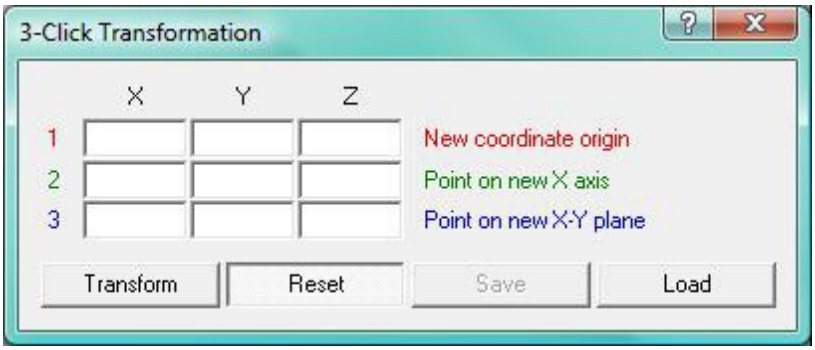
Note: This function is only available, if one of the result groups "shrinkage + warpage" is selected.

The 3-click transformation is just one of many possible procedures to temporarily define a transformed coordinate system. The transformation may include a transformation of the origin of the original coordinate system and a rotation of the coordinate axes.

(1) Select a shrinkage and warpage result.

(2) Click on “3-Click Transformation”.

⇒ The following window appears on the screen.



(3) Position the cursor in the first line.

(*See* Point on...function)

(4) Keep <Ctrl> pressed, and you select three points on your part, those you would like to assign the characteristics of the respective line (line inscription).

The cones can be stored only before the transformation.

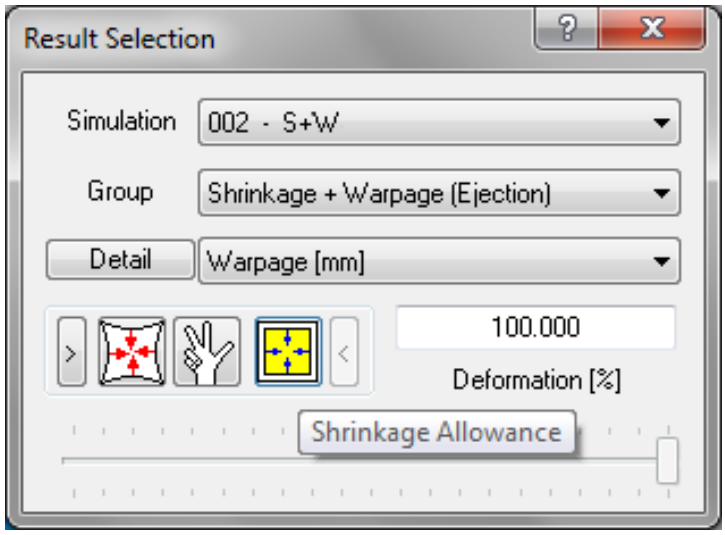
To compare results of different simulation variants it is important to have the equal conditions to each measurement.

In order to keep the conditions equal, the cones of the

"3-Click Transformation" can be stored and be loaded to measure a different simulation variant.

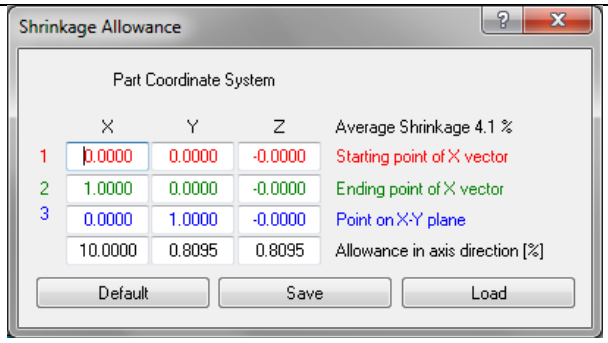
9.2.2.11 Shrinkage Allowance

The warpage-result (Result Group “Shrinkage + Warpage) can be calculated with anisotropic allowance. You can define the orientation of the new scaling system with entering 3 points (see 9.2.2.10 3-Click Transformation)



1. Select the result group “Shrinkage + Warpage”.
2. As detail “Warpage” must be chosen.
3. Click on the “Shrinkage Allowance”-Symbol.

→ The window “Shrinkage Allowance” appears.

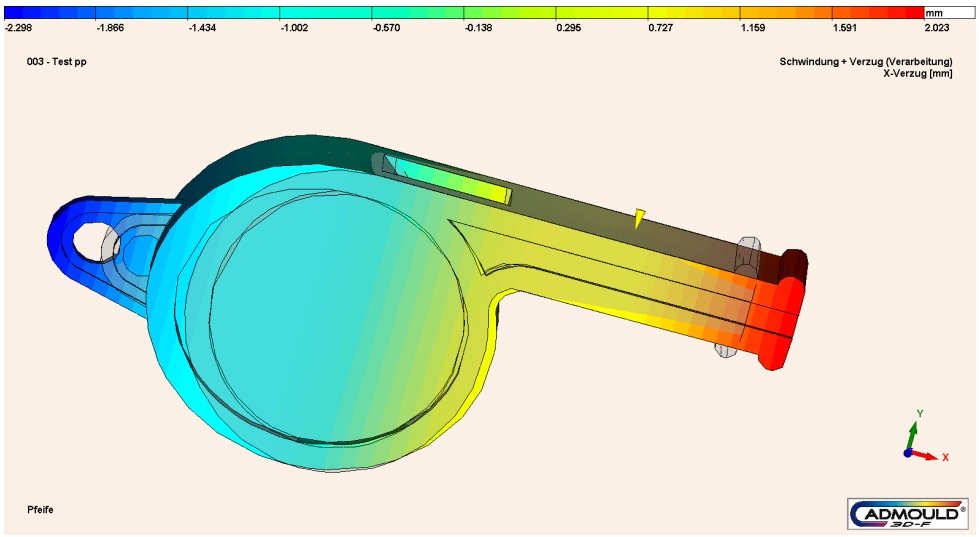


A description on how to enter coordinates see chapter 3-Click-Transformation

4. Enter a allowance in axial direction.

The "Default" button serves to set isotropic shrinkage allowance based on the average longitudinal shrinkage as a starting point..

For the following picture a X allowance of 10% has been chosen and 0.8% for the other directions.



As can seen quite clearly an anisotropic allowance (anisotropic stretching) is not angle preserving. Undercuts may arise and counter measures must be taken.

9.3 Interior results

In CADMOULD® 3D-F Version 3.50 you can now see results from the inside of the part. The results you can see are the following:

- residual stress
- filling temperature (interior)
- packing temperature (interior)
- filling velocity (interior)
- packing velocity (interior)

In each Simulation the interior results at the end of the filling- and cooling phase are stored.

If you want more results to be stored please consider the following steps:

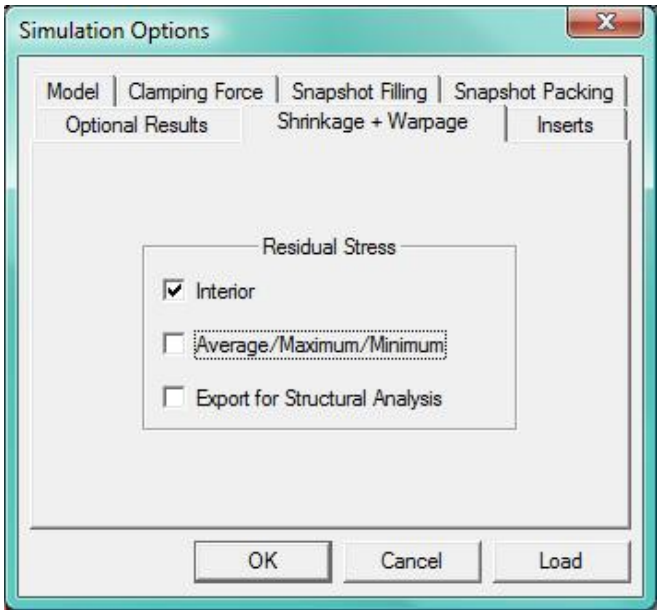
9.3.1 Activating the results

Open the window “Simulation Options” before you start the simulation.

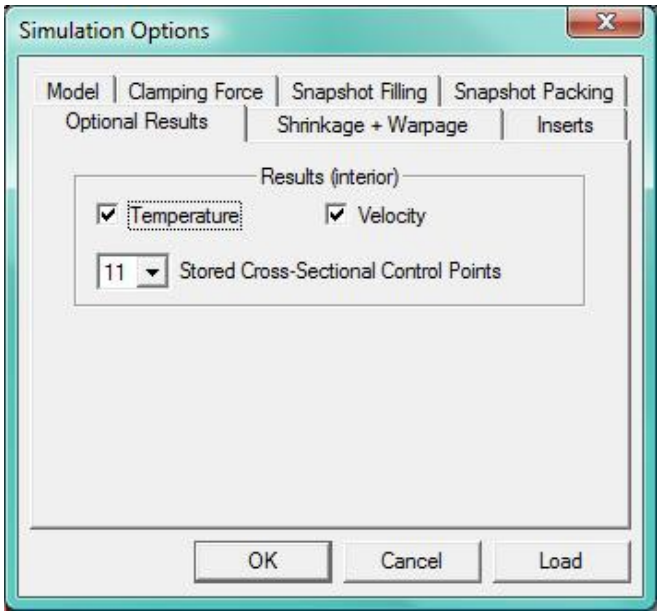
(Menu bar → Simulation → Options)

Jump to the tab “Shrinkage and Warpage”:

You can activate the result “Residual Stress – Interior” to see this result after the simulation.



Jump to the tab “Optional Results”:



Choose the interior results “Temperature” and “Velocity“ if you want to see them after the simulation for the whole process. The number of “Stored Cross-Sectional Control Points” allows you to adjust the resolution of the interior results.

Note: By default, CADMOULD® 3D-F saves the interior result at the end of the filling phase and the end of S+V.

9.3.2 Display of interior results

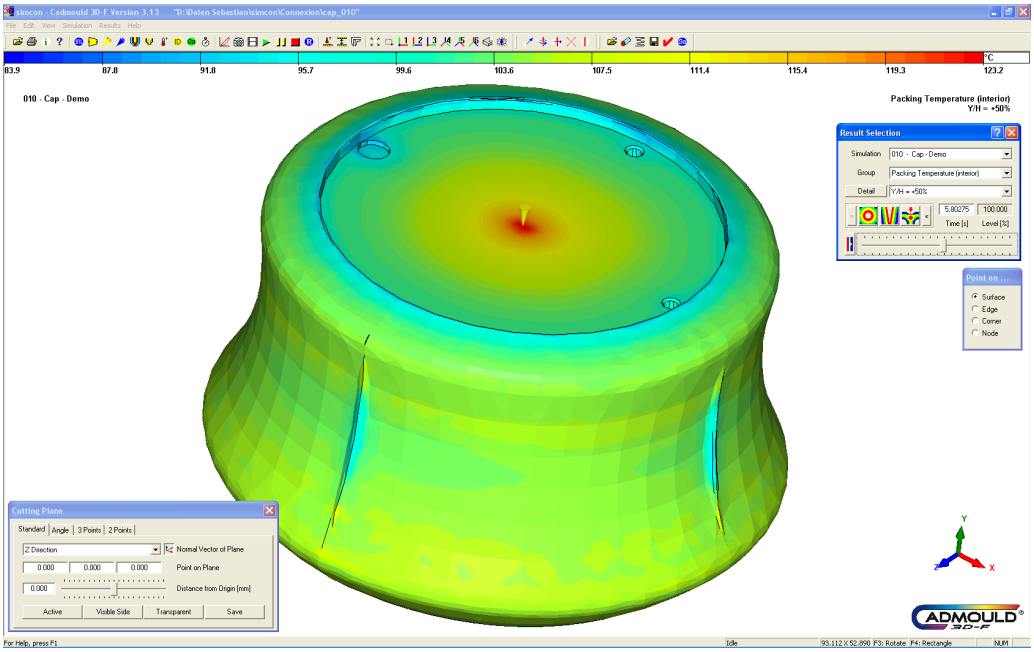
After executing a simulation with the settings as above you can now regard the interior results.

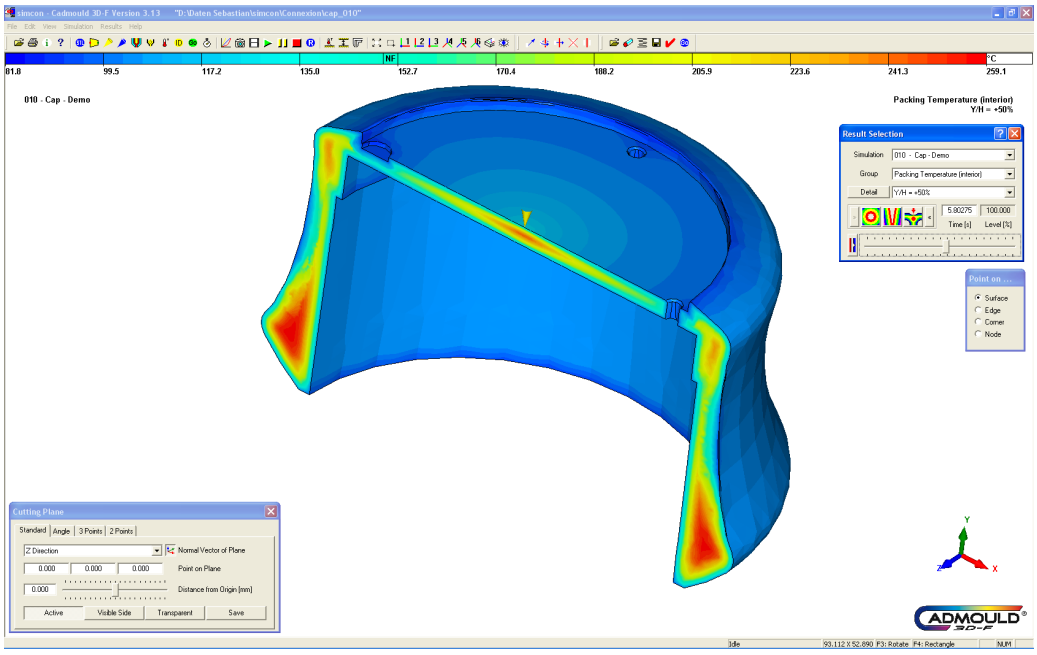
Open the result selection and select the group “Packing Temperature (interior)”.

Open the function “Cutting Plane” (Menu bar → View → Cutaway).

Activate now the cut by selecting a cutting direction and pushing the button “Active”.

You can now regard the interior temperature distribution at any point of time and in any cutting plane. You can also animate the temperature developing. Therefore you only have to push the play button (green arrow) in the button bar.





The result detail has no effect on the display of the interior results. The only effect is that the results of the chosen layer are projected on the shell of the part.

Here are some more examples of interior results:

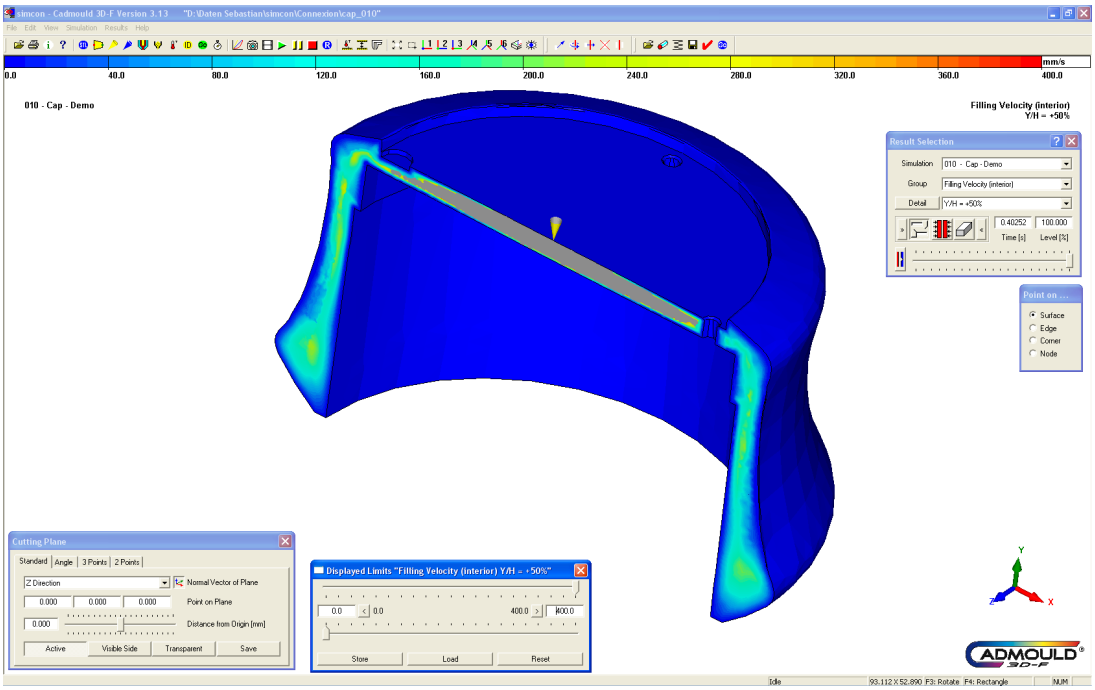


Figure: Filling Velocity (interior) with activated cutting plane

In the example above, the velocity was limited up to 400 mm/s to receive a better visibility of the differences.

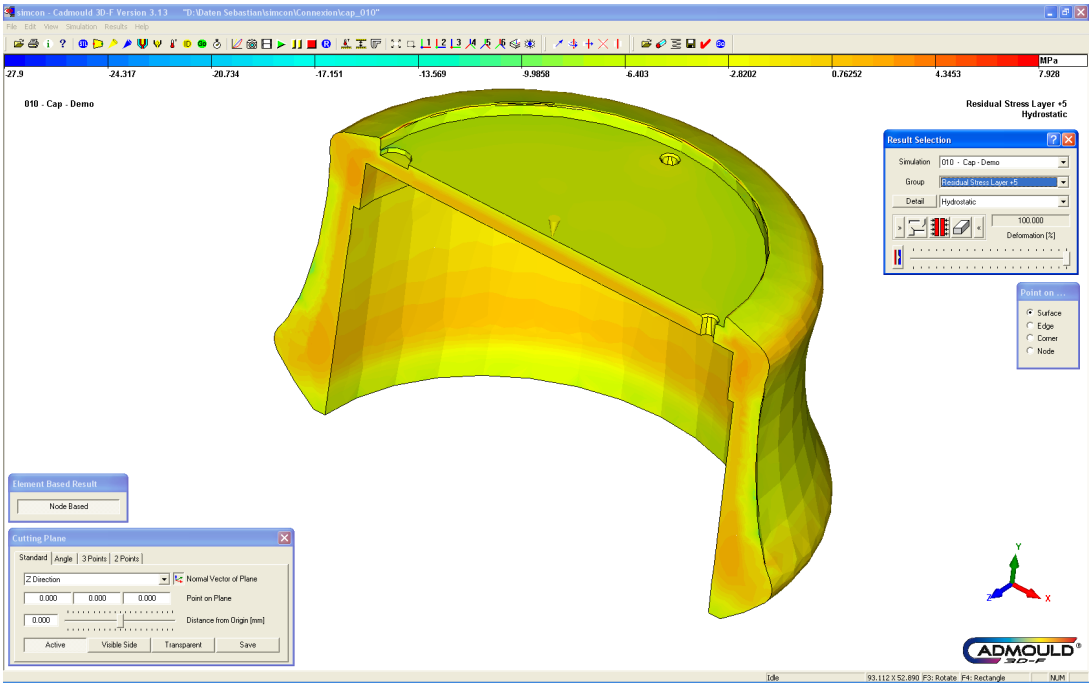


Figure: Residual Stress – Hydrostatic Node based with activated cutting plane

9.4 Animation of cross-sectional profiles

Since one of our last updates you can regard cross-sectional profiles for some results in CADMOULD® 3D-F. Now you can also animate these results in the form of a diagram.

Open therefore an existing project and choose the result “Temperature” from the result group “Snapshot Packing”.

Make sure that in the menu “Display Options“ on the tab “Results“ the function “Cross-Sectional Profiles” is activated.

Digitize now a point on the part with “Ctrl + left mouse button”. A diagram with the cross-sectional profile opens. You can now start the animation by pushing the play-button.

You can also jump to a point in time which you want to see with the slider in the window „Result Selection“.

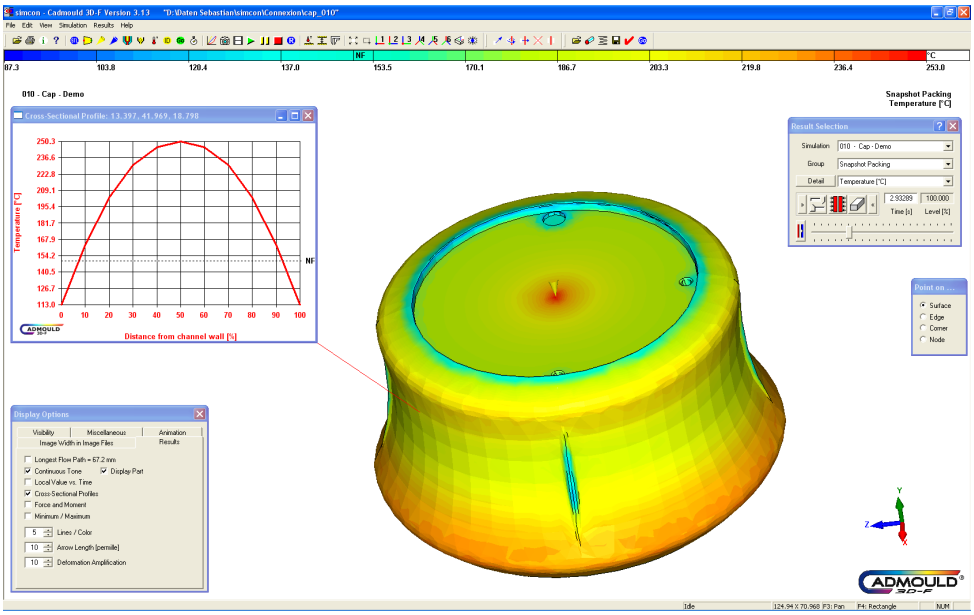
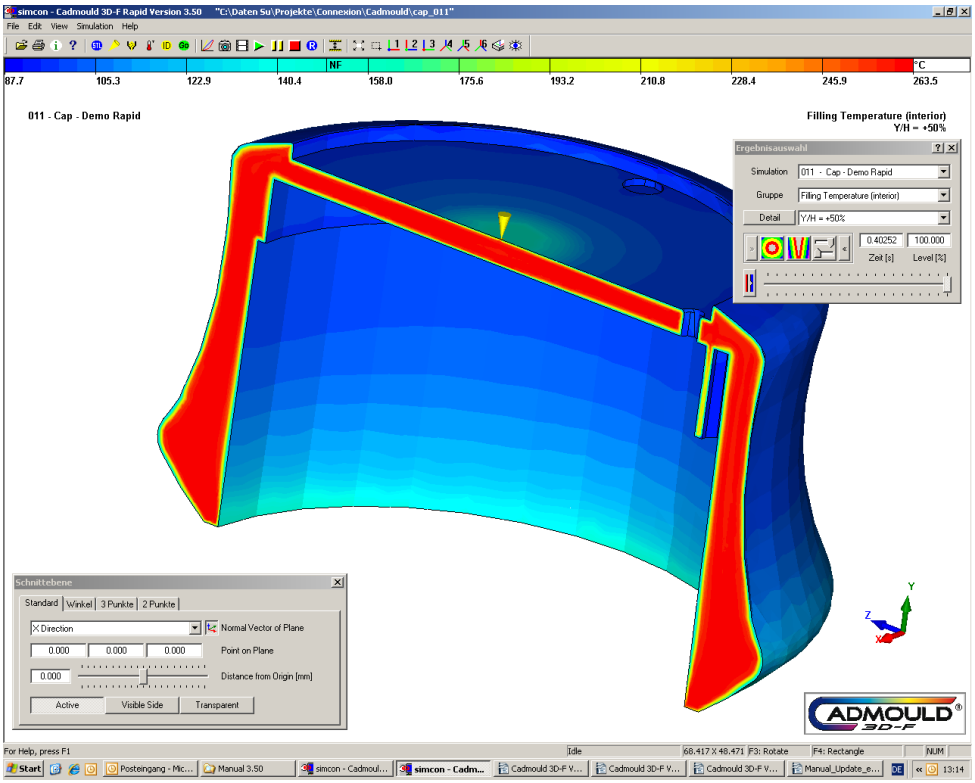


Figure: Cross-Sectional Profiles – Temperature Snapshot Packing



9.5 3D flow front

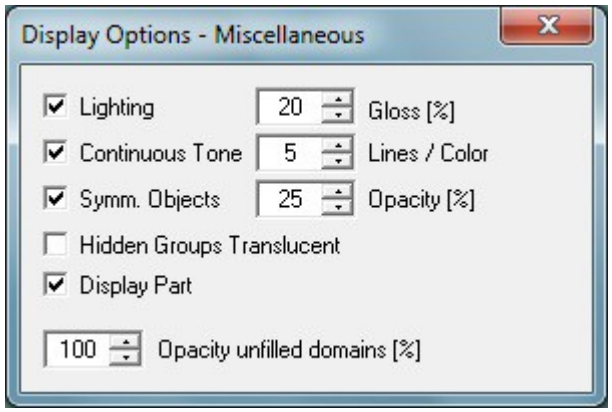
The latest version of Cadmould 3D-F gives you the possibility to see a 3D flow front.

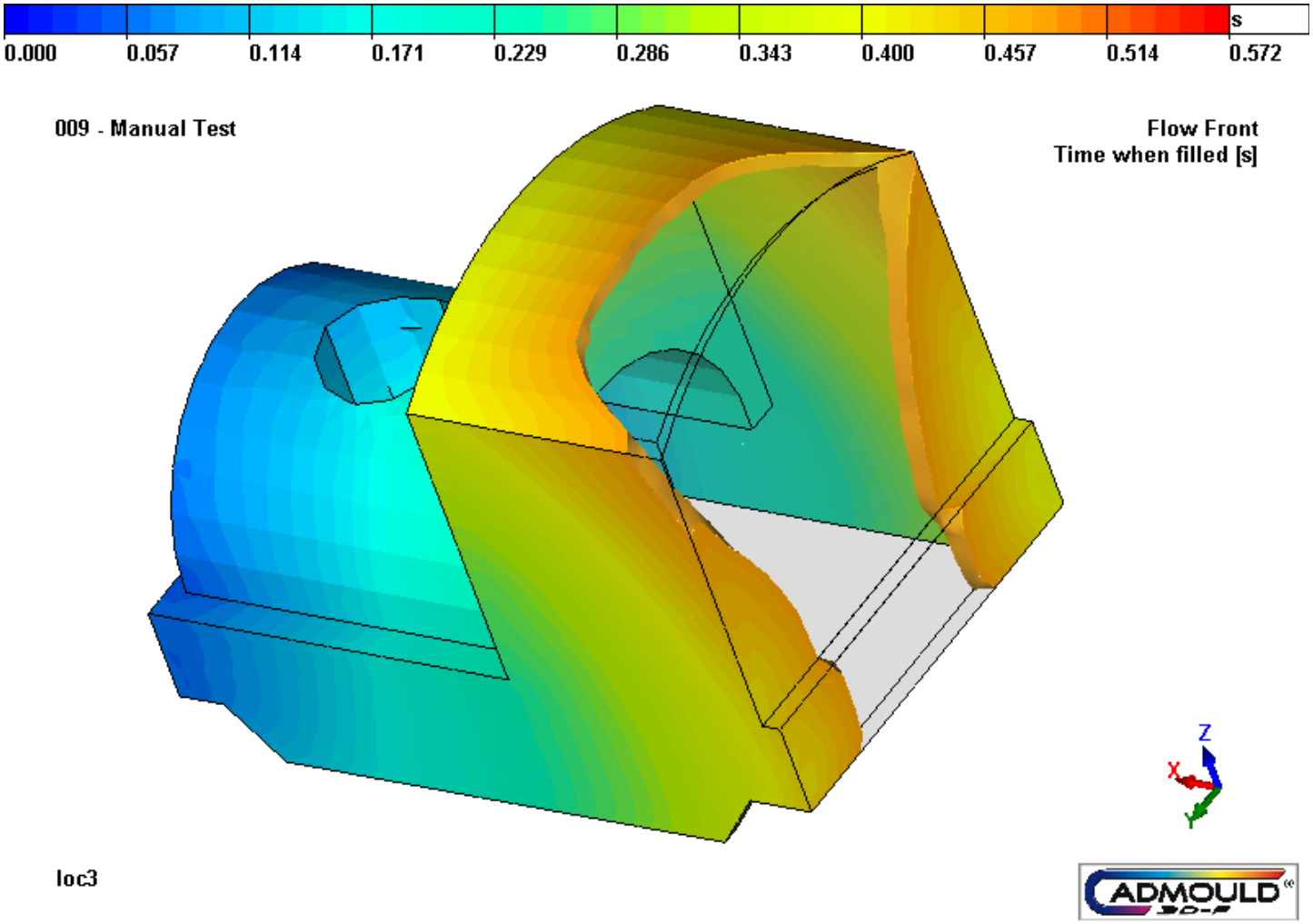
9.5.1 Activate the display of the 3D flowfront

Open the menu “View – Options – Miscellaneous”.

In the option “Transparency unfilled domains” choose a value smaller than 100%. A value about 25% is advisable.

Now display one of the results “Time when filled” or “Level when filled” from the result window and you get a 3D flow front when you move the flow front by slider or by entering a point in time.





Example of a 3D flowfront.

9.6 Evaluation of multiple results

If you have run several simulations for a part, you can select the simulation results in the window "Result Selection" under the title: "*Simulation*".

(1) Click the arrow in the line: *Simulation*.

(2) Select a simulation by clicking it.

9.7 Loading New Results

If you want to open a different part with already existing results, click "Open" in the pull down menu "File", or click on the first button in the toolbar.

(*See* chapter: “Open”)

10. Description of results

10.1 Result Summary

By choosing the group <Result Summary> a window is opened, which contains a table with the result developing over the time.
When you do a click on the title bar with the right mouse button, a configuration window opens. There you can for example adjust the order and the visibility of the column. There can also be loaded a standard configuration an own configurations can be saved and loaded.

10.2 Cavity

The geometry of your part without any result.

10.3 Flow Front

In <Time When Filled> and <Level When Filled> the filling pattern (isochrones) is stored. In <Time When Filled> the time is stored, at which each point is reached by the flow front.
In <Level When Filled> the filling levels is stored, at which each point is reached by the flow front.
Both filling patterns may be scaled using the slider and also may be animated.

In the ISO-Picture <Filled by Gate> you can see the influence of the different gates. The melt of each gate will be displayed in an own color.
In case of using cascade filling it is process conditional possible that the filling operation of following gates is unrealized.

<Temperature When Filled [°C]> holds the temperature of the melt at the time of each point being reached by the melt front, i.e. here the flow front temperatures are stored. The displayed temperatures are average values across the part's thickness.
<Pressure Loss [bar]> indicates the pressure loss from the point of injection to the respective place at the time of 100 % filling (the maximum if more than one injection point was defined.).

<Shear Stress When Filled [kPa]> contains the shear stress at the flow front. The maximum shear stress over the thickness usually occurring close to the wall or the frozen skin is given.

<Velocity When Filled [mm/s]> represents the flow front speed, when the flow front reaches the location you are looking at.

10.4 Snapshot Filling

The category <Snapshot Filling> contains snapshots, i.e. here values at specific times during filling are stored. The time can be set by the slide control in the result menu. In addition the results can be animated.
The following results are available:

<Temperature [°C]>:
The average temperature across the parts thickness (wall to wall).

<Pressure [bar]>:
The current pressure at each location of the part.

<Shear Stress [kPa]>:
The maximum shear stress across the thickness at each location of the part.

<Velocity [mm/s]>:
The average velocity across the parts thickness.

<Frozen Layer Thickness [%]>:
The thickness of the frozen layer as percentage of the parts thickness.

<Free X Section [mm]>:
The thickness of the flow able layer.

<Volume Shrinkage [%]>:
The volume shrinkage is the shrinkage potential of the plastics melt at a specific moment during filling and packing phase. The volume shrinkage contains beside the linear shrinkage also the gauge shrinkage. That's why the values can be comparatively high compared to the processing shrinkage. Volume shrinkage differences in the part causes different local shrinkage, what causes warpage.

10.5 Filling Orientation Degree

The average orientation degree can be displayed as an “iso image”. The values are between 0 (coincidental) and 1 (reliable). You can recognize, where isotropic and anisotropic areas are in the part. You can draw first conclusions about the stiffness and shrinkage behaviour from this information. Building the average of the layers is done from the orientation tensor. The average direction can be checked by activating all five fiber buttons. Instead of the colored fibers you can see one black fiber.

10.6 Force and Moment

The following results are displayed on the part:

- Resulting Force [kN]
- Moment Xn on object group [Nm]
- Moment Yn on object group [Nm]
- Moment Zn on object group [Nm]

The value of the respective result refers to the moment chosen in the result selection window. The development of force and moment can also be saved as animation.

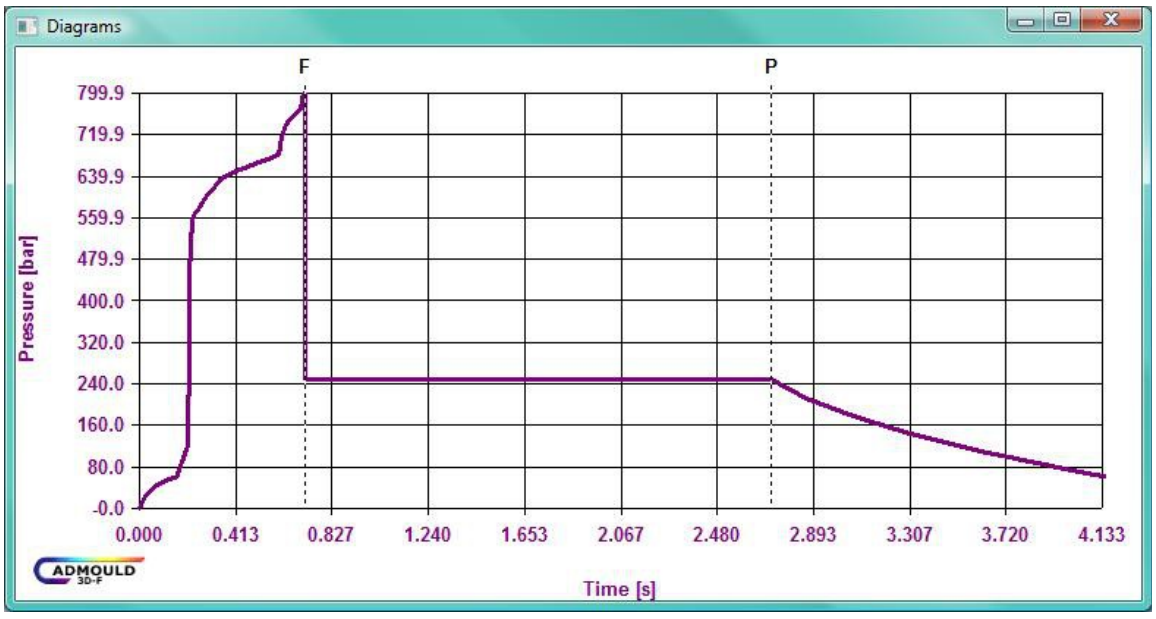
Note: The arrow of the respective moment is only displayed, if the moment has a certain influence on the result. Is for example the moment in X-direction negligible small, only the other moments are displayed.

(Please see also chapter: "Force and Moment (Display)")

10.7 Diagrams

This category contains nine diagrams:

- (Injection) Pressure [bar] vs. time [s]
- Flow Rate (at the gate) [cm^3/s] vs. time [s]
- Clamping Force X [kN] vs. time [s]
- Clamping Force Y [kN] vs. time [s]
- Clamping Force Z [kN] vs. time [s]
- Level (of filling) [%] vs. time [s]
- Mass [g] (contains also the runner system with hot runners)
- Mass - Gate [g]
- Mass Flow – Gate [g/s]
- Force Xn; Yn; Zn on object group [kN]
- Moment Xn; Yn; Zn on object group [kN]



Example: Pressure [bar] vs. time [s]

Note: When the option "Local Value vs. Time" in the menu View, Options, Results is activated, there are more diagrams available.

(See chapter: "Local Value vs. Time")

10.7.1 Diagram Functions

Each result of the diagram can be probed in the diagram window by clicking it with “Ctrl/Strg + left mouse button”. In the left top corner of the diagram window the result at the desired location is displayed.

If you load a diagram curve from a CAR file also the abscissa can be selected freely. Default is "Time [s]".

Zoom curves without input via keyboard: You can simply zoom the diagrams with a rectangular window without selecting any other option before. If the curves are zoomed, there is an option in the context menu (right mouse button) "Unzoom curves" to show the complete curves. Alternative option is the shortcut "Ctrl/Strg+F”.

Note: There is a line connecting the diagrams "Mass – Gate n [g]" and "Mass Flow – Gate n [g/s]" with runner segment n.

10.7.2 Functions in the Context Menu

Click with the right mouse button on the diagram to call the context menu.

10.7.2.1 Limits

With this function you can fit the display area of the X- and Y-Axis.

10.7.2.2 Division

In this function you can set the number and the distance of the divisions from the X- and Y-Axis.

10.7.2.3 Display of the Curve

The function “Font” allows it to change the font of the labeling.

In the function “Background Color” you can choose the color of the diagram background.

If you use the function “Text Color (permanent)”, the text color will be always as set. By choose function “Text Color = Curve Color”, the text color will always be the same like the curve color.

In the functions “Curve Color” and “Pen Width” you can set the display options of each curve.

10.7.2.4 Delete Curve

With this function you can delete several curves of the diagram.

10.7.2.5 Curve to the bottom

With this function you can sort the curve to the bottom of the list on the right hand side.

10.7.2.6 Scale Like Active Curve

With this function you can scale the ordinate of not homonymous curves like “Clamping Force X”, “Clamping Force Y” and “Clamping Force Z”.

10.7.2.7 Smooth Curve

Here you can choose values from 1 (no smoothing) until 1000 (smoothing function) to set the smoothing of the curve.

10.7.2.8 Maximize Curve

With this function you can choose, which curve is scaled that it contacts the bottom and the head of the diagram.

10.7.2.9 Rename Curve

To look at two homonymous curves of different projects, you can rename them here.

10.7.2.10 Curve => Table

This function compiles a table with the values of the curve. This saves the way over “File – Convert – Car → Text”.

10.7.2.11 Load Curves

Here you can load curves in the actual diagram in different ways.

10.7.2.11.1 From File

This function serves to load curves from any other Diagram-file produced by CADMOULD® 3D-F, also of different projects.

10.7.2.11.2 From Clipboard

Diagram-Curves can also be load from the clipboard.
The content of the clipboard can have 2 different formats:
a) A text with data pairs like this:

5	1363
10	1377
15	1397

or

5	1363	10	1377	15	1397

This function can read decimals, points and commas without problems: "1" = "1.00" = "1,00" Additional you can mark a table in MSExcel and copy the content with Strg/Ctrl+C to the clip-board.

b) A second possibility is to load a bitmap from the clipboard. For this you need a program which can copy images to the clipboard. Any image editor should be able to do this. It is possible to load cutaways from an image as well. If it is not possible to load an image, the reason can be, that there is too much useless image information in the image. In this case you can erase these labels,

legend or scale using eg MSPaint. If there are gaps in the curve, CADMOULD® 3D-F cannot read this as one curve. You have to stitch it with a few pixels first (in your image editor).

To decide between the curves, CADMOULD® 3D-F needs different colors. Possibly you have to repair that, too.

10.7.2.12 Variation

This function allows you to load the bar charts or compare curves of an variation, if performed.

10.7.2.13 Store Image File

With this function you can save the diagram window as PNG- or BMP-file.

10.8 Residual Stress

Avg. Hydrostatic

Max. Von Mises

This results can also be displayed as diagram over the profile. Digitize <Ctrl + left mouse button> a point on the part to see the diagram.

10.9 Part Quality

Points which are difficult to fill can be recognized with the result <Filling Problems>.

There are four categories:

green	=	good to fill
yellow	=	problems with the filling are likely
red	=	there are problems with the filling
white	=	not filled

Criteria are comparison of flow front and melt temperature as well as comparison of filling time and sealing time.

For the analysis of <Sink Marks> the sealing time is compared with the time of freezing.

The <Ejection Time [s]> is the time when the **average** temperature reaches the ejection temperature.

<Sealing Time [s]> means every connection to a gate is frozen.

<Freezing Time [s]> means the time at which the **entire** cross section has reached the no flow temperature.

The <Gradient Volume Shrinkage> describes the quality of the sink marks. It´s hundred of times the gradient of the thickness shrinkage. The gradient expresses, how much the reflected light is

distorted, respectively casts a cloud. A high value describes a great changing of the surface direction in comparison to the design. If the part is comparative planar at this point, you can see a sink mark or a buckle there. By our experiences this is the case, if the value is higher then 3.

10.10 Snapshot Packing

See chapter: "Snapshot Filling"

Note: During the remaining cooling time, CADMOULD® 3D-F will continue writing 111-files. The packing time will be extended up to the ejection time.

10.11 Shrinkage & Warpage - Cooling

The max. and min. temperature occuring in the mold [°C] over the time [s].

10.12 Shrinkage & Warpage (Ejection)

Results describe the state outside the mold immediately after ejection from the mold.

This category comprises these results:

- X-Deformation [mm]
- Y-Deformation [mm]
- Z-Deformation [mm]
- Deformation [mm]
- Shrinkage [%]
- X-Warpage [mm]
- Y-Warpage [mm]
- Z-Warpage [mm]
- Warpage [mm]
- Curvature Change [%]

The first 4 deformation results are the components and the overall values of deformation based on shrinkage **and** warpage.

The <shrinkage> is the local linear change in distance in the part, occurring in the surface extension. It is not directed, i.e. directional differences as they are typical for fiber reinforced materials are displayed as average values. The mean shrinkage of the entire part is given in the status line.

The <warpage> is the overall deformation reduced by the above defined mean shrinkage.

Except the local shrinkage all deformation results depend on the position of the warped part and the original one. For the standard the mean deformation is minimized. The relative position can be changed using the **3-Click Transformation**.

The <curvature change> is used by curved faces and free formed surfaces, which could not be measured normally. A positive result describes a convex deformation, a negative result describes a concave deformation.

10.13 Shrinkage & Warpage (Final)

The results show the state outside the mold after cooling down to the ambient temperature entirely.

This category holds the same types of results as the category “Ejection” described above.

10.14 Interior

At the interior results, a result is assigned to every node depending on the time. These result distribution can also be displayed as a diagram over the profile or on cutting planes.

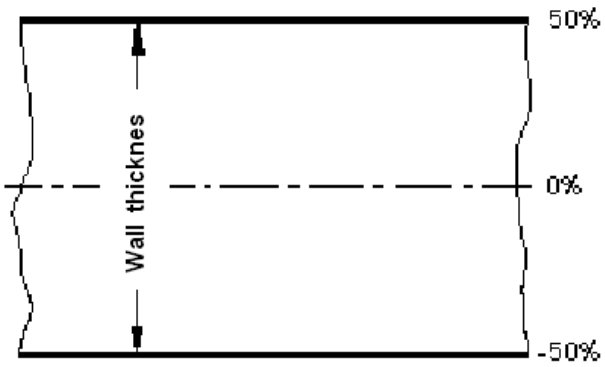
To see this diagram, you have to digitize a point on the part with <Ctrl + left mouse button>.

The following interior results are available:

- Filling Temperature
- Filling Velocity
- Packing Temperature
- Packing Velocity

The wall thickness of the part is divided into 10 layers. That means 10+1 check points. Time-dependent-temperature or -velocity is assigned to each check point.

Please see also chapter: Cross-Sectional Profiles



10.15 Residual Stress Interior

The remaining Hydrostatic Stress and the remaining Von Mises Stress after cooling to ambient temperature. These results can also be displayed as cross-sectional profiles. Digitize a point on the surface of the part to show this profile.

10.16 Wall Temperature Analysis

The max. and min. temperature at the mold [°C] over the time [s], as well as the inlet and outlet temperature and the flow rate for each cooling circuit can be displayed in a diagram.

10.17 Heating/Cooling System

The <Pressure [bar]> describes the pressure at each point of the System.

The <Reynolds number [1]> (Re) is a dimensionless number, which describes the ratio between the inertia and viscosity forces. For the pipe stream accounted here, it is:

Re = average speed * diameter / kinematic viscosity of the fluid
If the Re number decreases below a critical threshold of approx. 2300, the favored turbulent flow switches to a laminar flow, which can dissipate much less heat from the cooling wall. The Re number and further typical numbers are necessary for the calculation of the heat flow to the coolant fluid.
Because of obstructions and fluctuations are not allowed to have influence on the heat flow, the Re number should be high enough over the threshold. With rising value there is a repletion of the impact, so that an increase produces nearly no advancement, but requires a higher pump capacity.

The <Heat transfer coefficient [W/(m²K)]> describes the temperature transfer between the mold and the coolant fluid.

The <Heat flow efficiency [1]> shows if an element takes surpassing or substandard much of the whole heat supplied. The whole length of the system is accounted.
E<1 The element absorbs substandard much heat
E=1 The element absorbs average much heat
E>1 The element absorbs surpassing much heat
That means, that the system is balanced designed, if all elements have the efficiency 1.

The <Temperature [°C]> shows, where and how the temperature of the temperate medium arises. A coolant bore, whose medium heats much, dissipate much heat and is perhaps too small dimensioned.

10.18 Sensor

The category <Sensor> contains five diagrams:

- Pressure [bar] vs. time [s]
- Temperature [°C] vs. time [s]
- Viscosity [Pa·s] vs. time [s]
- Shear Rate [1/s] vs. time [s]
- Specific Volume [cm³/g] vs. time [s]

Check-Point-Diagrams and Check-Points are connected by a line.

Each result of the diagram can be probed in the diagram window by clicking it. In the left top corner of the diagram window the result at the desired location is displayed.

11. Variation Results

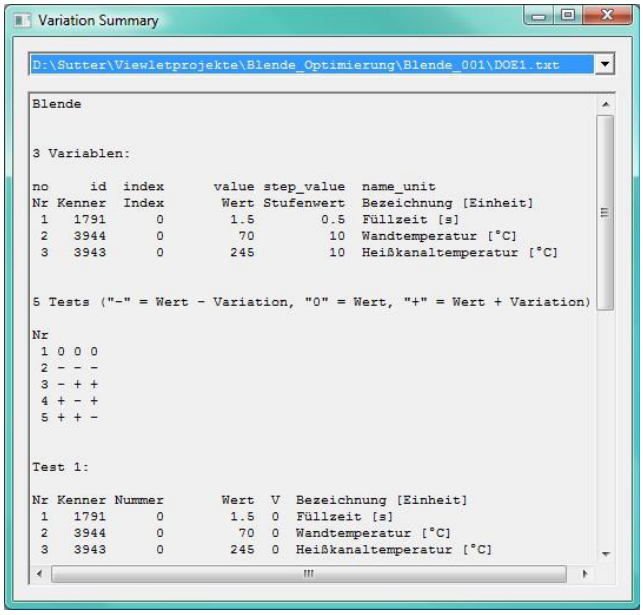
If you've used the "Vary" function in the dialogue "Process Parameter" or another variation method (DOE), you can see the results after the simulation in the menu "Results → Variation".

Note: The variant calculations are only made automatically, if the module “Batch” is licensed. Otherwise you have to start all necessary variant simulations manually.

11.1 Summary

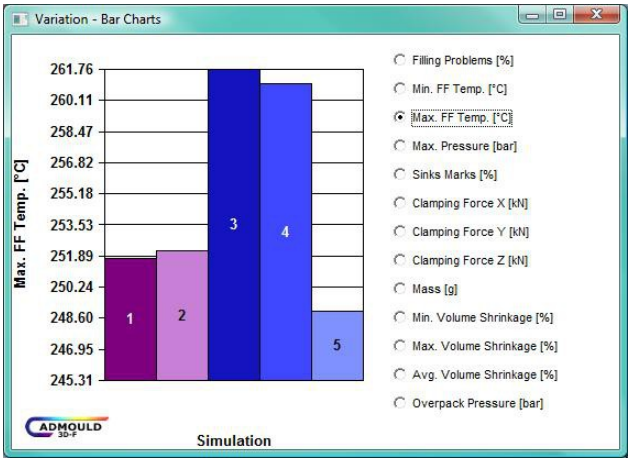
When you choose the function "Summary", the window "Variation Summary" appears on the screen.

In this window you can see, which parameters are varied and which process parameters has which simulation ID.

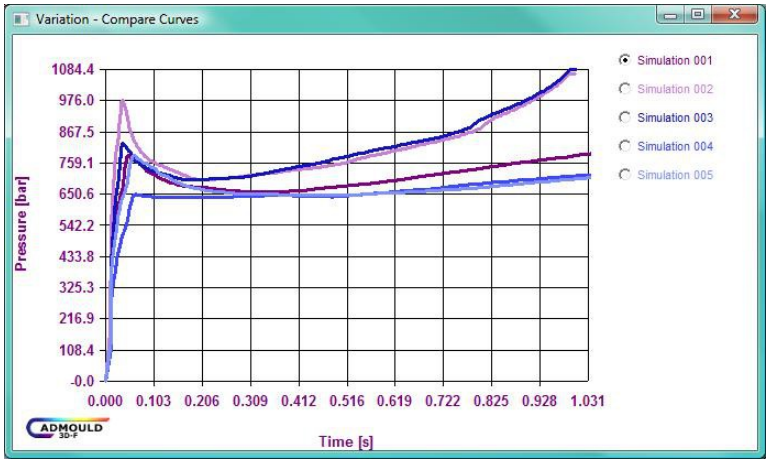

Variation Summary
D:\Sutter\Viewletprojekte\Blende_Optimierung\Blende_001\DOE1.txt
Blende
3 Variablen:
no id index value step_value name_unit
Nr Kenner Index Wert Stufenwert Bezeichnung [Einheit]
1 1791 0 1.5 0.5 Füllzeit [s]
2 3944 0 70 10 Wandtemperatur [°C]
3 3943 0 245 10 Heißkanaltemperatur [°C]
5 Tests ("-" = Wert - Variation, "0" = Wert, "+" = Wert + Variation)
Nr
1 0 0 0
2 - - -
3 - + +
4 + - +
5 + + -
Test 1:
Nr Kenner Nummer Wert V Bezeichnung [Einheit]
1 1791 0 1.5 0 Füllzeit [s]
2 3944 0 70 0 Wandtemperatur [°C]
3 3943 0 245 0 Heißkanaltemperatur [°C]

11.2 Bar Charts

When you choose the function "Bar Charts", the following variation bar charts appears on the screen. On the right hand side, you can choose, which result you want to display. The numbers in the bars accord to the simulation ID.



11.3 Compare Curves



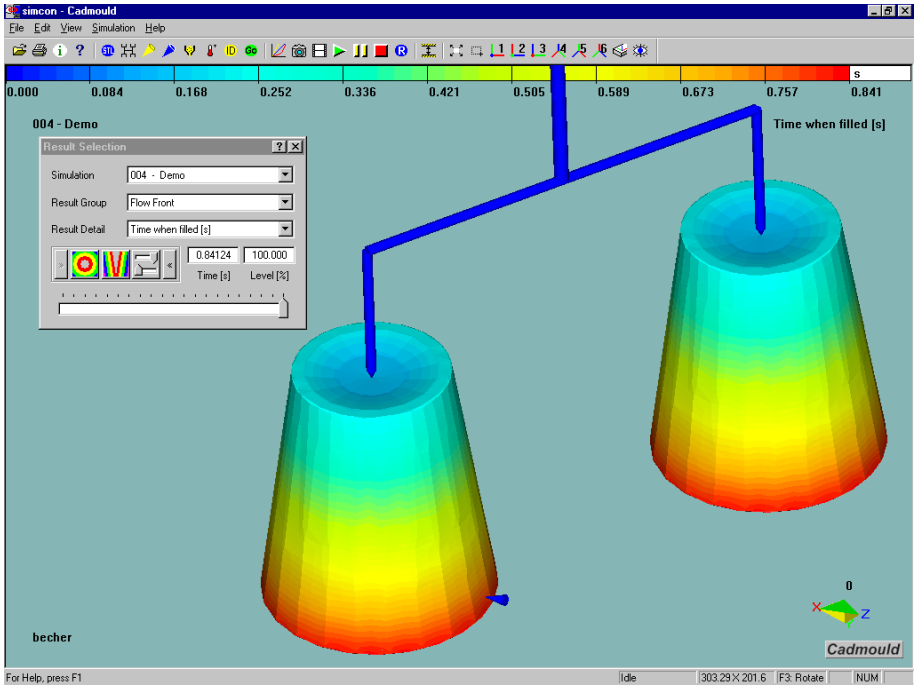
When you choose the function "Compare Curves", a diagram appears on the screen, where you at first have to choose the value for the X- and Y-axis in a separate window.

On the right hand side, you can choose to which simulation the axis will be scaled.

12. Examples of the results

You find the result: "Time When Filled" in the result category "Flow Front“. This result shows isochrones of the flow fronts during filling. Each color is assigned to an equal long time range depending on the number of colors in the scale.

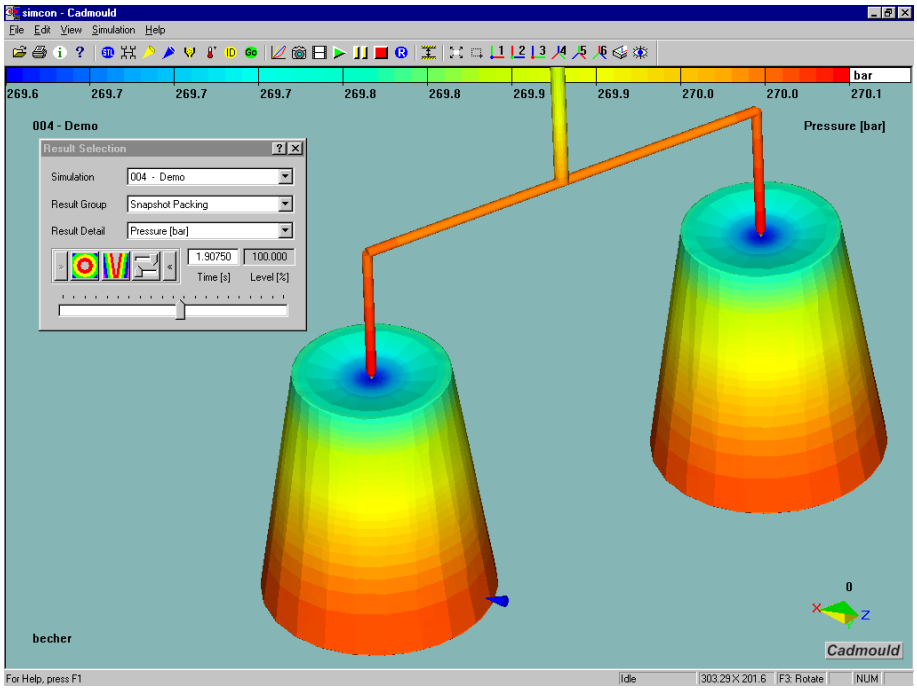
e.g.: the color blue is assigned to the time range 0.000 – 0.0168s, the time 0.83s is shown in the last red color that encloses 0.824 – 0.841s.



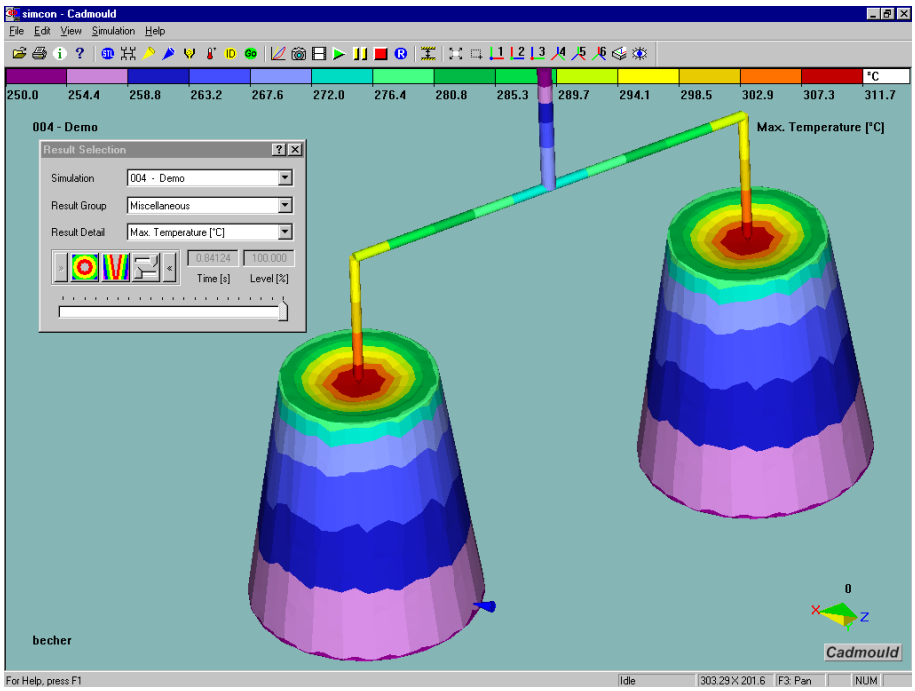
You can detect potential problems due to weld lines or air traps in this presentation.

(*See* chapter: "Weld Lines", "Air Traps")

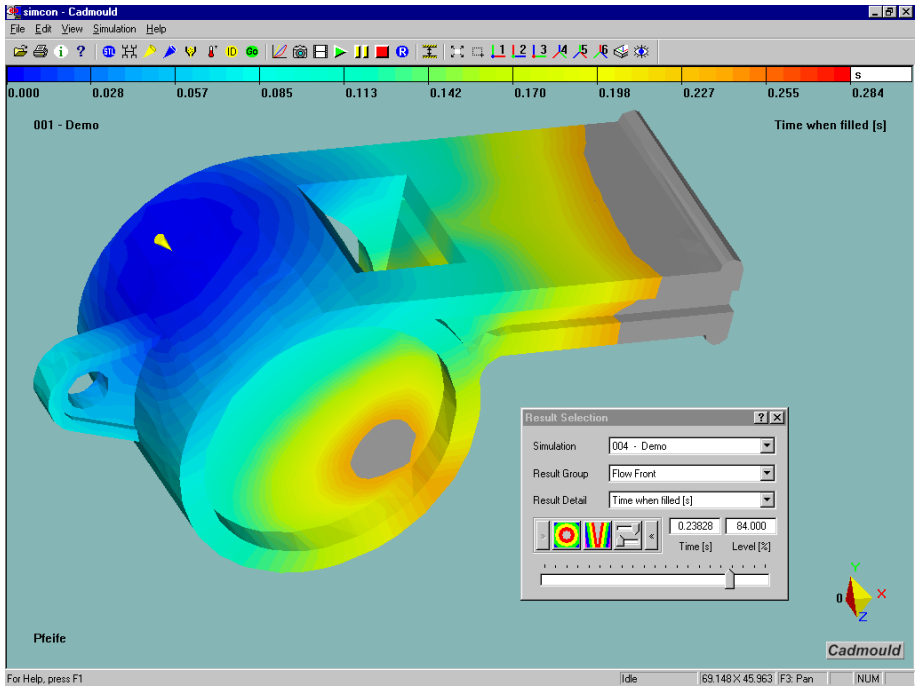
This picture shows the current packing pressure at each point of the part. The time can be set either with the slider or by numeric input in the time field above the slider. For filling snapshots there is the additional choice of numeric input of the filling level.



The diagram shows the maximum temperature at the end of the filling phase for each place / point at the part. (Color scale set to 14 colors for better reading)

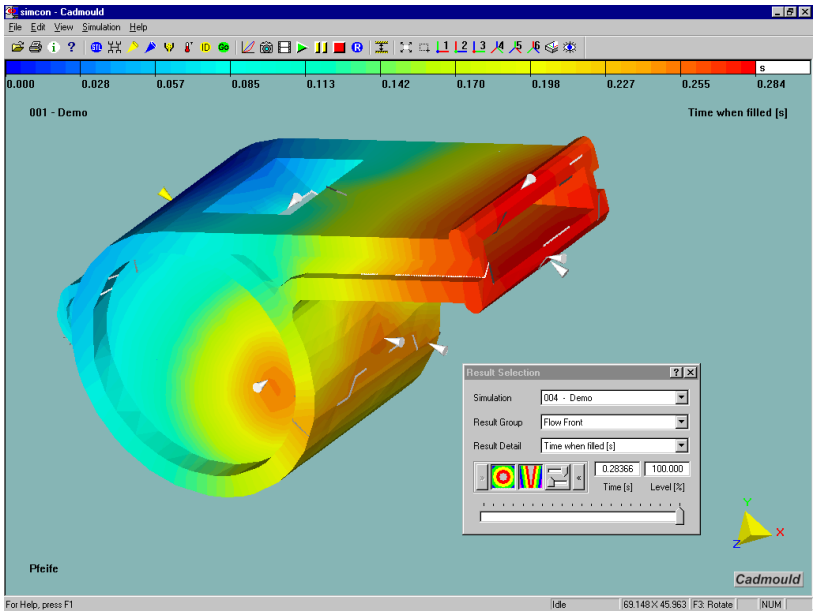


A filling simulation of a whistle was run to show the following results.

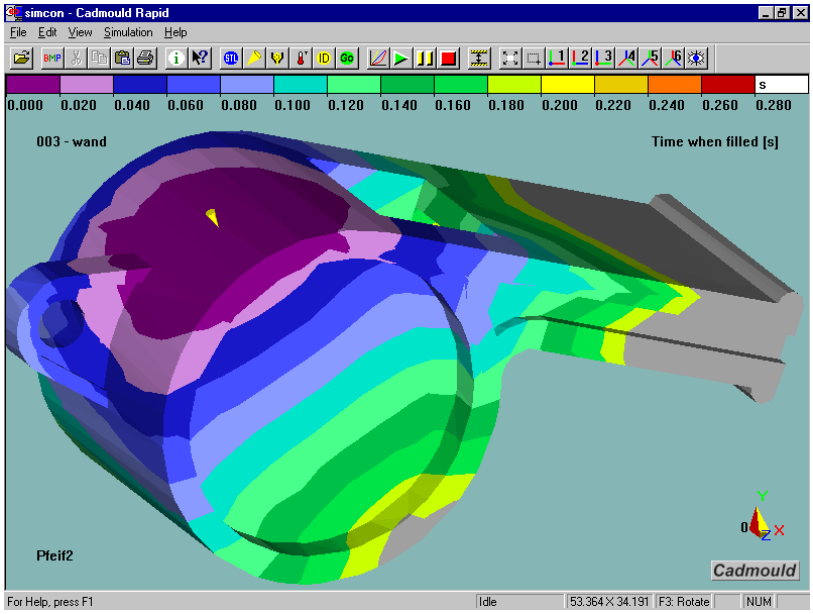


The result shows an air trap at the round wall. An air trap can lead to the so-called “Diesel effect” and burn the material. Obviously, the flow front is slower in the round wall than the flow front in the cylinder wall, this is the reason for the air trap. In order to increase the flow front speed in the round wall, the wall thickness is changed. (*See* chapter: “Thickness”)

The air traps and weld lines can be indicated.



The next picture shows the result of a simulation, after the wall thickness was changed from 1mm to 1.5 mm. The air trap at the round wall is avoided.

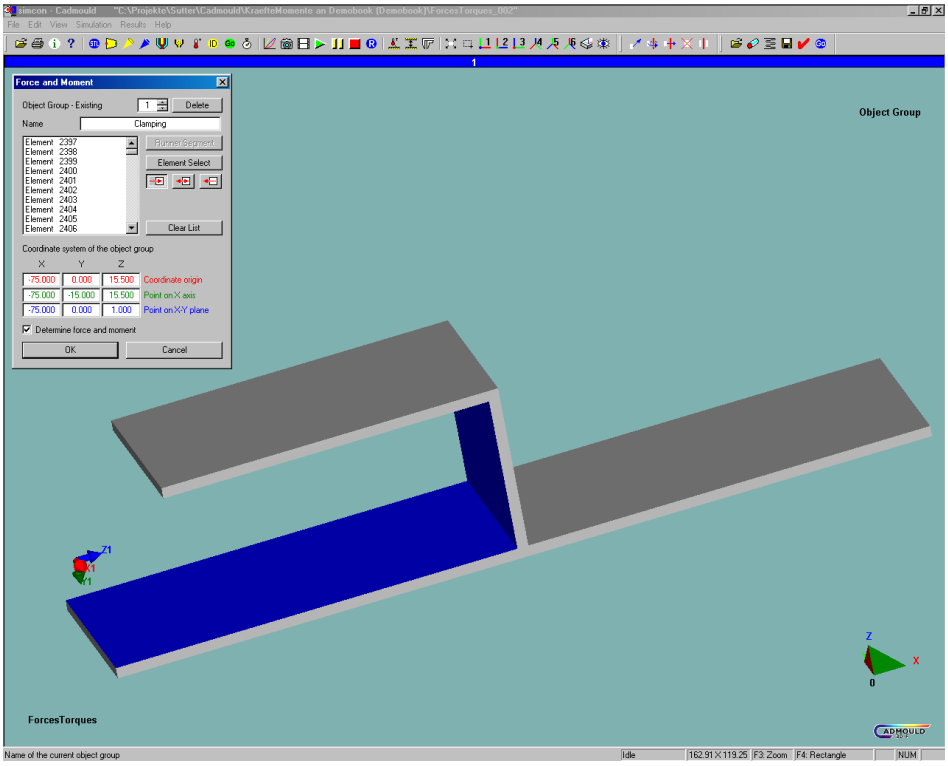


12.1 Example: Force and Moment

In the following example, the force and the moment on the slider in the clamping should be calculated.

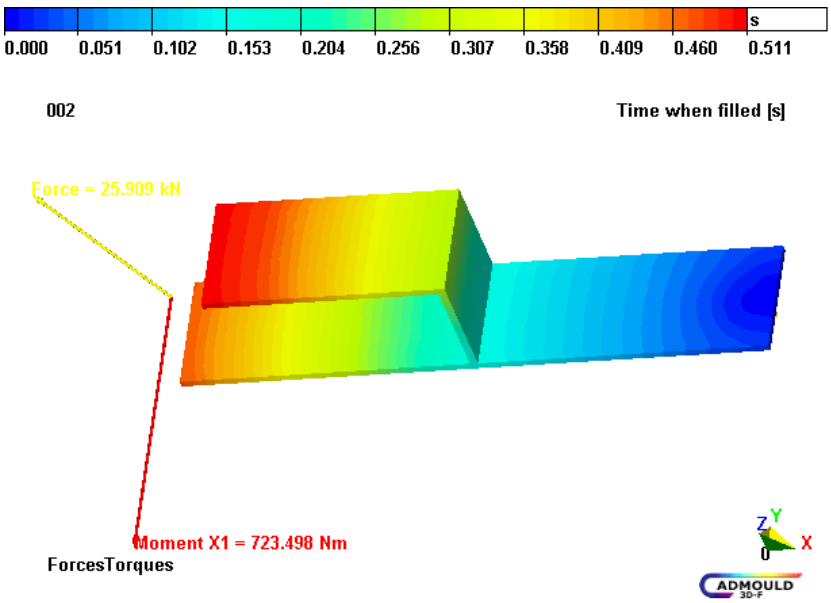
(See also chapter "Force and Moment (View)" and "Force and Moment (Simulation)")

This picture shows the cavity with the selected inner surfaces and the defined coordinate system. the upper inner surface is also selected, but because of perspective reasons not visible.



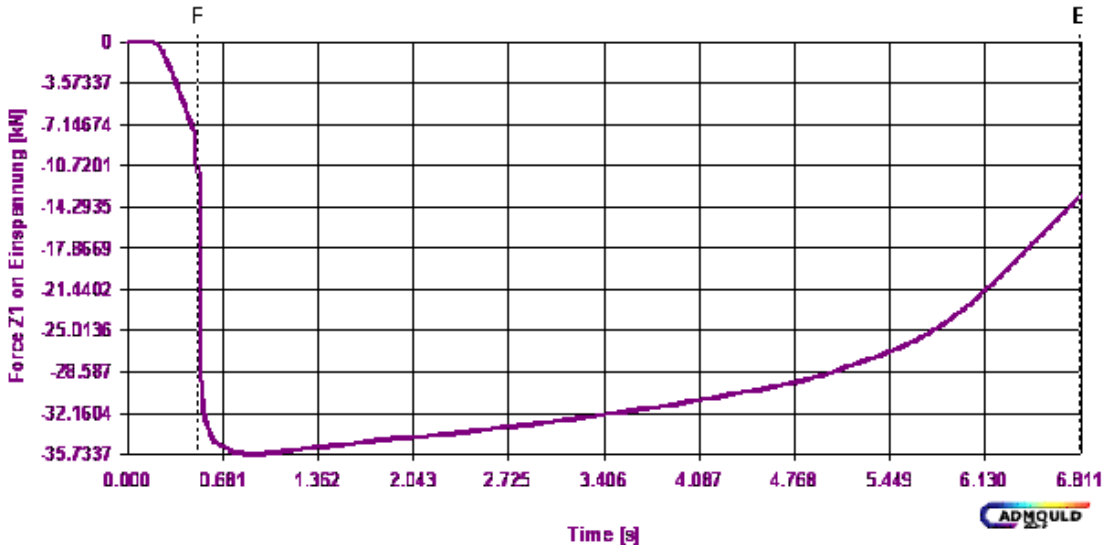
In the following pictures you see the display of the results:

Picture 1: Result of the calculation of force and moment



Note: Only the moment in X-direction is displayed, because the others are negligible low. As result of the forces is only displayed the resulting force.

Picture 2: Force in Z-direction over the time



You can see this diagram for all three directions, as also for the moments in all three directions.

(See also chapters: "Force and moment (Description of results)" and "Diagrams")

13. Animations

13.1 Online Animations

You can animate all results in the categories: "Snapshot Filling", "Snapshot Packing" and the results: "Time When Filled" and "Level When Filled" in the category "Flow Front".
(*see* chapter: "Description of results")

For playing an animation use the keys "**Play**", "**Pause**" and "**Stop**" or use the button in the menu bar.
(*see* chapter: "Button Bar")
Before you play an animation consider please the chapter:
"Results, Animation Frames"

13.2 Single-Step Frames

In the window "Result Selection" you can look at single results in each stage of the injection molding process. You can determine the time and the filling level by keyboard or mouse.

⇒ by keyboard

♦ define the time

(1) Mark the field "Time [s]".
(2) Enter a time value.
(3) Press "Return".

♦ define the filling level

(1) Mark the field "Level [%]".
(2) Enter the filling level between 0 - 100.
(3) Press "Return".

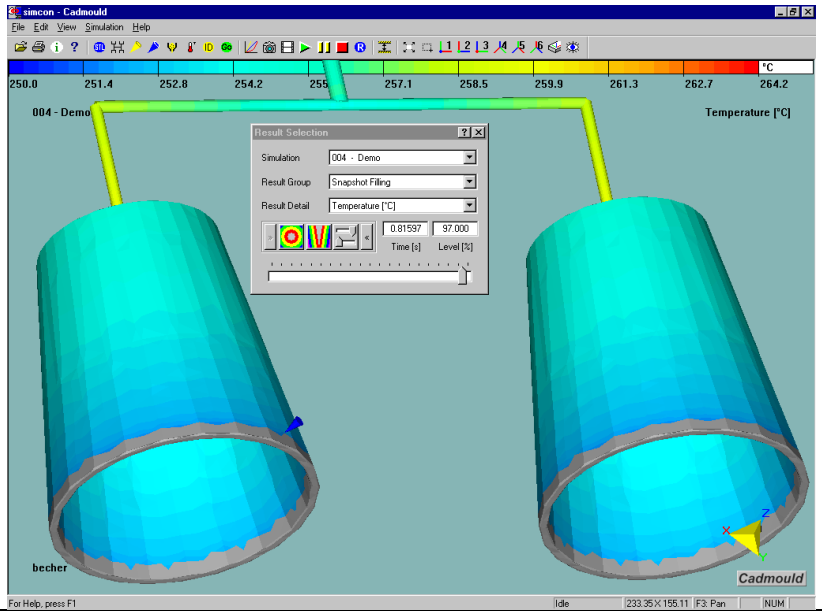
⇒ adjust the slider

♦ by mouse

(1) Drag the slider. Resting for a short period will let the display adjust to the time position.

♦ by keyboard

(1) Position the cursor on the slider
(2) Shift the slider using the ←/→ keys.



Example: 1

Picture shows the average temperature at a filling level of 97%.

13.3 Store Animations

You can store every animation as an avi-file. You can insert these into your reports, presentations etc. and run it with the appropriate software, e.g. Microsoft's Media Player. It is not possible to view the stored animations with CADMOULD® 3D-F.

Alternatively animations can also be saved as a sequence of pictures. Therefore you can choose BMP or PNG as file format.
The result is a sequence of picture-files like this example:

"Movie_00000000.bmp"
"Movie_00000001.bmp"
"Movie_00000002.bmp"
...

The function "File -> Convert -> BMP/PNG -> AVI" allows you to convert such a sequence into an AVI-animation.
Please see chapter: "Convert"

Before storing a film please consider the chapter: "Animation".

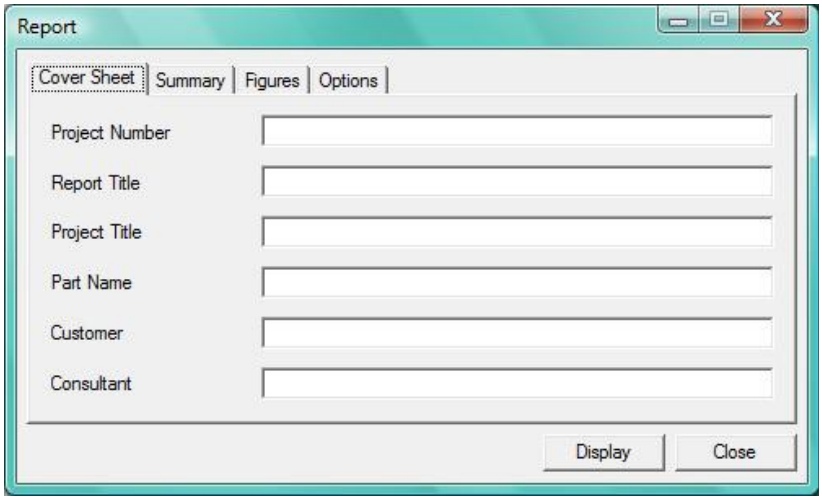
1) Select a result an animation can be run on.
2) Click "Animation File".
   ⇒ The window "Save as" appears on the screen.
3) Give the path, the directory and a name for the file.
4) Choose the file format. (AVI for an animation or BMP/PNG for a sequence of single pictures)
5) Click "Save".

14. Report Filling Analysis

A user defined report of your filling simulations can be generated automatically by the function “report”. You can print this file or send it as an e-mail.

(1) Click “Report”.

⇒ The window “Report” appears on the screen.



(2) Enter your project data in the template "Cover Sheet" completely.

(3) In the template "Summary" add a short summary to the report.

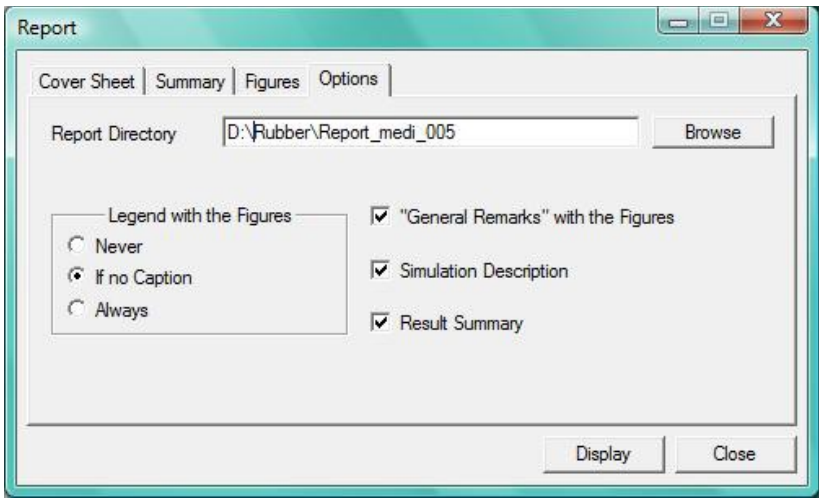
(4) Click “Default” in the template "Figures".

⇒ The window "Open" appears on the screen.

Several RIL-files are in the directory where you have installed CADMOULD® 3D-F. Each RIL-file stands for a language e.g. ger.ril stands for German, eng.ril for English, ther-ger.ril for thermoplastic - German, elas-ger.ril for rubber - German.

(5) Open the RIL-file of your choice. All figures defined therein are generated automatically. The current aspect of your part will be applied, so set this before you select the report file.

(6) Determine the form of your report and its administration in the submenu "Options".



Appearance: Mark how to proceed with the legend of the figures. The legend is a set of information about the type of result and the process that the program generates automatically. It can be controlled depending on the presence of your caption.

Mark "General Remarks" to integrated these into the report. General remark may be helpful for people not so familiar in interpreting simulation results. They also give some guidelines how to improve possible defects.

Administration: By creating a report a subdirectory is created where all files belonging to the report files are stored. The report itself can be stored in a separate directory.

Mark whether you would like to store the report in a subdirectory or in the report-specific directory.

The report-specific directory can be selected with the browser or be created manually by indicating a new path,

e.g. D:\Reports\John

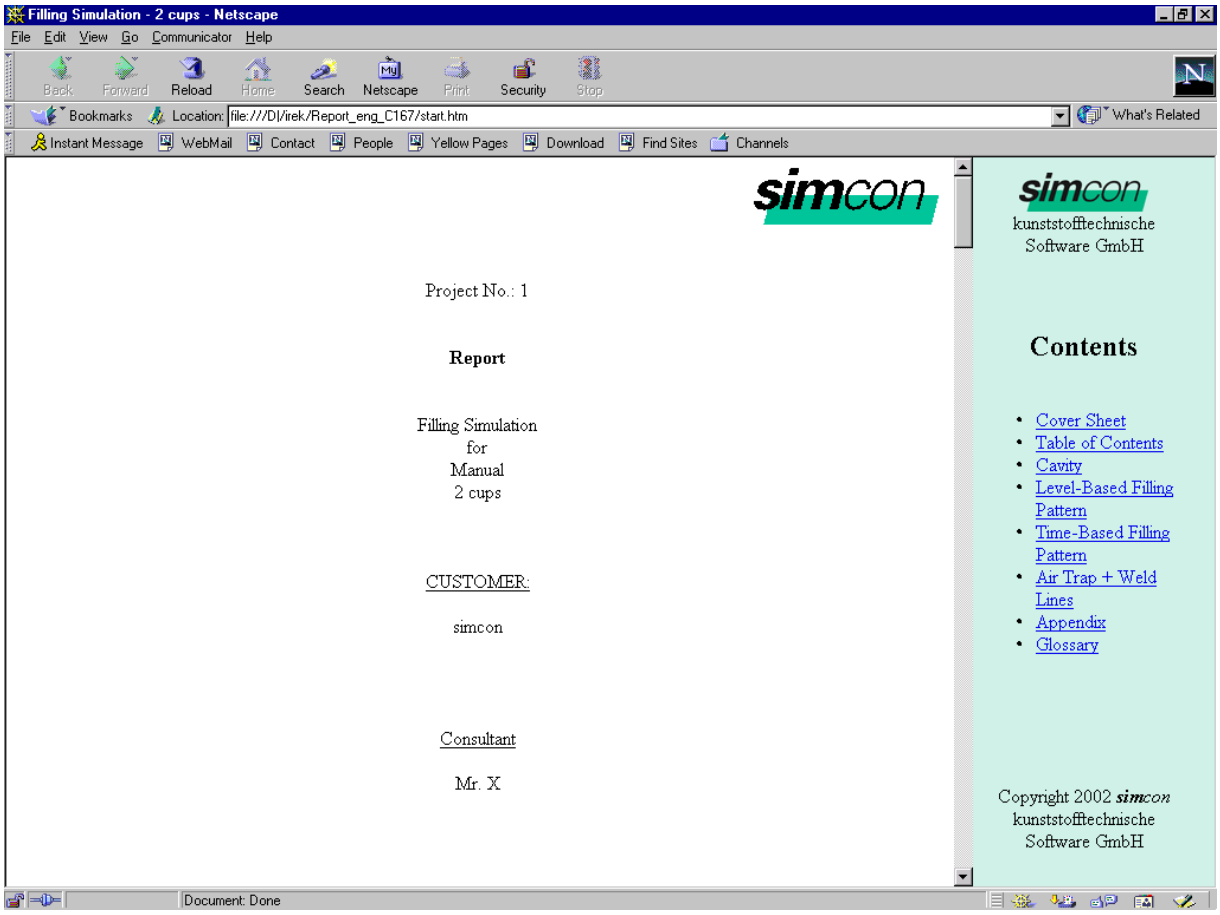
In this case the directory "John" is created.

(7) Click “OK”. Clicking “Apply” instead allows previewing the report and incorporating changes before finalizing the report with “OK”.

If there is already a report, and you want to create a second report of the same filling analysis, you have to change the directory where it should be saved, or rename the existing report directory in the explorer.

The report is provided in an htm-file format and is opened in an Internet browser .The name of the report is the same as the name of the title of your report. The report is stored in the directory you have defined in the template “Options”.

(8) Click on "Yes" in order to open the report now.



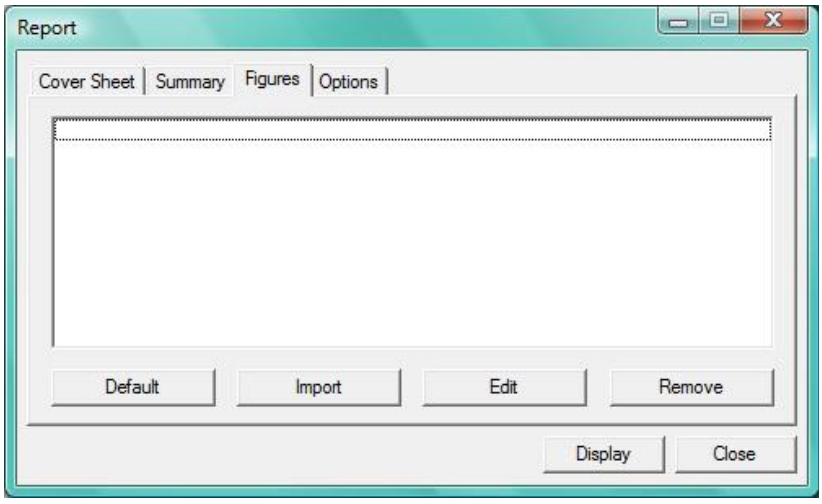
14.1 Define a Report Standard

You can provide different report templates (RIL-files). Therefore it is possible of create different reports automatically, based on user-defined report templates.

(1) Activate the template "Figures" in the window "report" in order to create a report template.

(2) Click "Default".

⇒ The window "Open" appears on the screen.



In the directory CADMOULD® 3D-F was installed to, there are several RIL-files. Each RIL-ile stands for a language, e.g. ger.ril stands for German, eng.ril for English, ther-ger.ril for thermoplastic – German, elas-ger.ril for rubber - German.

(3) Open the appropriate RIL-file in order to automatically generate result pictures of your part.

⇒ CADMOULD® 3D-F provides a standard report. You can edit and store the report. Use the functions "Edit", “Import” and "Remove" in the window "Report" in the template "Figures", in order to modify the report.

(4) After the editing of the report click on "OK ".

CADMOULD® 3D-F creates a RIL-file, which stores the appearance of your report. This file is stored in the same directory as your RM1-file of your part. So you can create further reports off this report.

You can provide several report templates, integrate texts and pictures into your reports. Use the functions "Import", "Edit" and "Remove" in the window "Report" in the template "Figures".

Notice: If you provide several report templates of the same part, then you have to rename the pertinent RIL-file after each newly created report.

14.2 Import Results

You can integrate your own result pictures or Simcon-standard-report pictures into your reports.

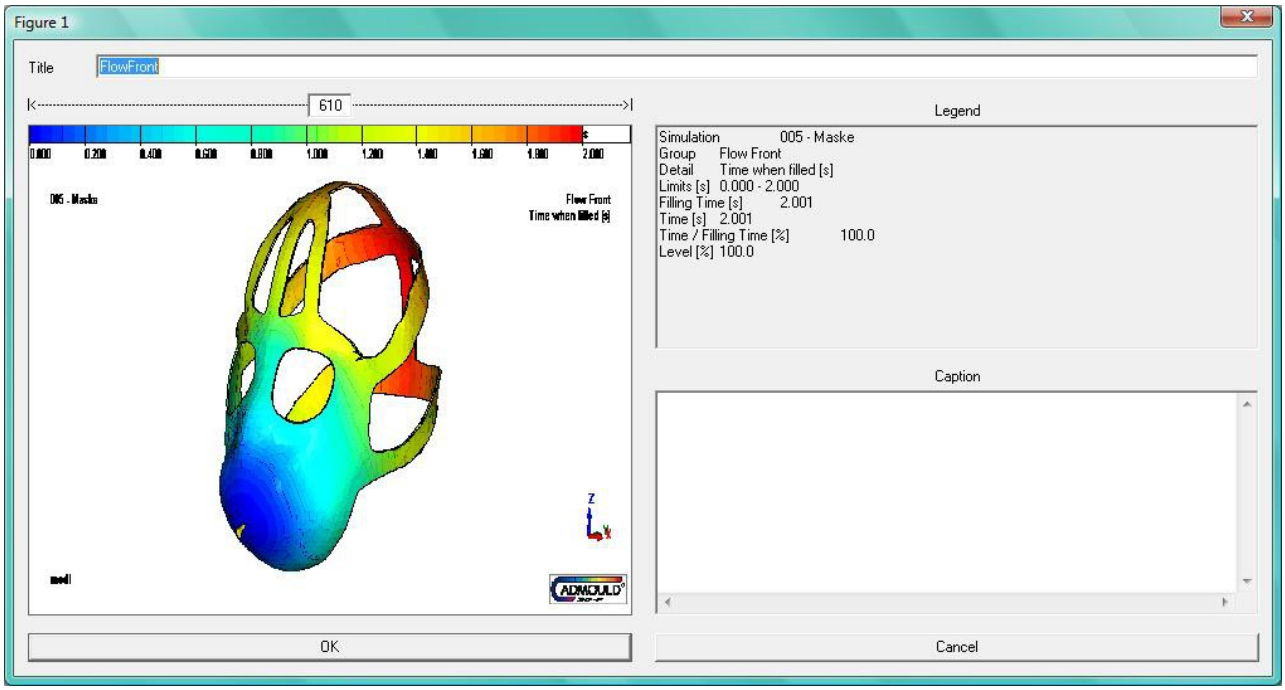
(*See* chapter: "Save Image")

(1) Click "Import".

⇒ The window "Open" appears on the screen.

(2) Mark an image file and click "Open".

⇒ The following window appears on the screen.



(3) Enter a title and a comment in the fields "Title" and “Caption”. Then click “OK”.

14.3 Edit a Report

You can edit your results in the report with the function "Edit".

Mark a result and click on "Edit".

⇒ The selected result can be edited in the window, like you see before.

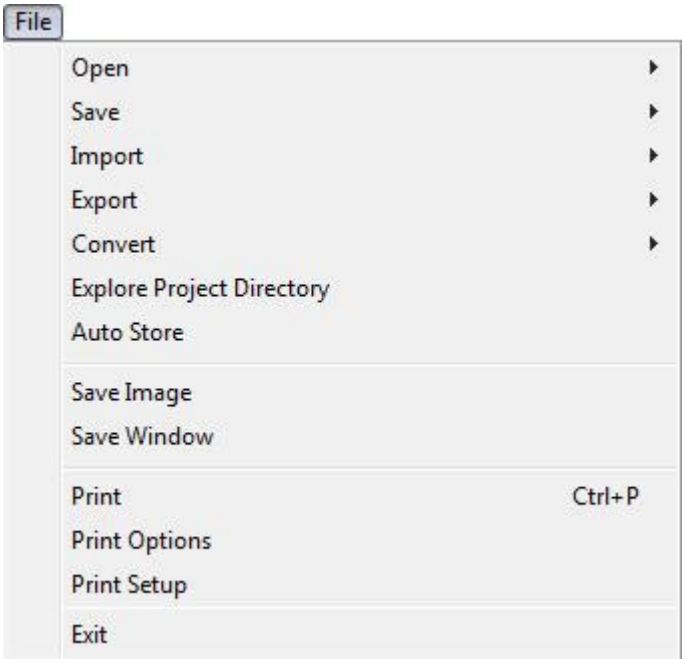
15. CADMOULD® 3D-F Viewer

CADMOULD® 3D-F ***Viewer*** is a program of the ***sim****con* company, which serves for the representation of all CADMOULD® 3D-F simulation results. Additionally HTML project reports can be generated automatically.

(*See* Chapter: Report Filling Analysis)

In case of questions about a license, please contact your distributor.

16. Menu Bar



16.1 File

Click "File".

⇒ The pull down menu "File" is activated.

In the pull down menu "File" you find instructions for handling the projects.

16.1.1 Open

16.1.1.1 Project

The window "Open" appears on the screen. You can load an already existing result “RM1-file”, symmetry file “SYM-file” or a design file “DES-file”.

RM1-files contain all information of a project.

SYM-files (symmetry files) are project files. They contain origin geometry (e.g. STL file), all duplications of origin geometry and the runner system.

DES-files are project files. They are automatically created after you have determined a geometry title and a geometry identity number in the window "Simulations Description".

(1) Search the results of your part.

(2) Click on the file.

(3) Click on "Open".

(4) Wait until the geometry of the part is loaded and appears on the screen.

16.1.1.2 Runner System

Already existing runner systems (RUN – Files) can be loaded or merged. Already loaded runner systems can be replaced by a new one.

Alternatively runner systems can be imported as centerlines in IGES format. During the import you are prompted to set a general diameter for all segments.

16.1.1.3 H/C System

Already existing heating/cooling systems (HCS – files) can be loaded or merged. Already loaded heating/cooling systems can be replaced by a new one.

Alternatively cooling channels can be imported as center lines in IGES format. During the import you are prompted to set a general diameter for all channels.

In addition h/c systems in stl file format can be loaded and converted into the Cadmould 3D-F format.

Please see for STL h/c system import chapter: 3D Heating/Cooling Channel

16.1.1.4 CADMOULD® 3D-F Any Result (CAR)

You can load *.CAR files of any result on any part, as long as the number of nodes resp. the number of elements is the same. This function serves to display newer results in general.

16.1.2 Save

16.1.2.1 Project

Projects can be stored in the RM1 file format.

(1) Click “Save Project”.

⇒ The window "Save as" appears on the screen.

(2) Enter the name of the project.

(3) Click "Save".

16.1.2.2 H/C System

Heating/Cooling systems with particular or several cycles can be stored. You can choose between three different file formats:

- HCS - file format (H/C system file)
- IGES - file format
- CFE - file format (CADMOULD® 3D-F FE-mesh file)

(1) Click “Save H/C System”.

⇒ The window "Save as" appears on the screen.

(2) Enter the name of the heating/cooling system.

(3) Click "Save".

Note: Saving h/c-systems as IGES - files will save no diameters. At the beginning of the IGES - file you find the following table, from which you can see the allocation between the stored numbers and the type of the segment.

N	colour	Segment
2	red	regular
3	green	baffle
4	blue	bubbler
5	yellow	hose

16.1.2.3 Runner System

Runner systems can be stored in the following formats:

- RUN - file (Runner system file)
- IGES - file

(1) Click “Save Runner System”.

⇒ The window "Save as" appears on the screen.

(2) Enter the name of the runner system.

(3) Click "Save".

Note: Saving runner systems as IGES - files will save no diameters. At the beginning of the IGES - file you find the following table, from which you can see the allocation between the stored numbers and the type of segment.

N	Colour	Profile
2	red	cycle
3	green	annular shape
4	blue	parable
5	yellow	square
6	magenta	trapezoid
7	cyan	rounded square
8	white	virtual segment

16.1.3 Import

16.1.3.1 Mesh Files

CADMOULD® 3D-F can read files in these formats:

ANSYS, Cadmould-FE, Cosmos GFM, Fillcalc, Hypermesh ASCII, I-DEAS, NASTRAN, PATRAN, Raw and STL

For some of these formats you require a separate license.

Example: Import of STL (this is always available)

(1) Click "**STL**", in order to load a new geometry.

⇒ The window "Open" appears on the screen.

(2) Search the STL-File (stereo lithography file) of your part.

(3) Select the file.

(4) Click "Open".

⇒ The window "Scale the part to millimeters" appears on the screen.

(5) Chose a scaling factor regarding the dimensions.

(6) Click "OK".

(7) Wait until the geometry of the part is loaded and appears on the screen.

Note for the Import of FE-mesh-files: This option is only available in CADMOULD® 3D-F FE products. Proceed in the same manner as described for **STL**.

You need to add the extension .cfe to the file name to import a CFE-file you have already exported by CADMOULD® 3D-F or used in ***cmv6***. The restriction to four characters no longer exists.

(4 characters + .cfe = file name)

e.g. bech + .cfe = bech.cfe

16.1.3.2 Material Data

With the function "**Material Data**" you can assign a material to current projects.

(1) Click on „Material Data".

(2) Open the appropriate plb file (*Material*.plb), or you can import the Material Data from an another projetct. Therefore you have to select the project file (*.rm1), which contains the material data.

⇒ The material is assigned to the current project.

(*See* chapter: Material Data, i0 Interface)

16.1.3.3 Wall Temperatures

CADMOULD® 3D-F can proceed the distribution of the wall temperatures. Files of the wall temperature distribution (Project_ *identity-number*_**TMP0**) can be imported into projects of the same FE-mesh.

Proceeding: Import different wall temperatures into a current project.

1. Enter the "Title and the identity number of the project".

(Gate, material and the process parameters should be already entered. In the field "Wall Temperature" in the window "Process Parameters" you can enter a default value at first.

2. Load or define a Heating/Cooling System.

(*See* chapter:Toolbar "Wall Temperature",Import a heating/cooling system)

3. Define the heating/cooling circuits.

(*See* chapter:Heating/Cooling Circuits)

4. Define the coolant fluid and the mold material.

(*See* chapter: Options – Wall Temperature Analysis)

5. Exit the Window: "**Options – Wall Temperature Analysis**" by clicking "OK".

6. Start the Wall Temperature Analysis.

(*See* chapter: Wall Temperature Analysis)

The created file of wall temperature distribution (project_*identity-number*_**TMP0) can be imported into the current project.**

(*See* chapter: "Wall Temperature distribution)

The mean wall temperatures of the filling and cooling phase can be indicated element- and node-referred in the window "Wall Temperature [°C]".

Click on "Wall Temperature".

16.1.4 Export

16.1.4.1 CADMOULD® 3D-F FE

You can export the “Cadmould FE Mesh” of you actual project. In this file the FE-mesh of all parts and if present also all runner- and h/c segments is saved. In case that the mesh isn't already prepared this will happen during the export of the file.

(1) Click “File → Export → Cadmould-FE”.

⇒ The window: “Name of the mesh (4 characters)?” appears on the screen.

(2) Enter a name of the mesh using exactly 4 characters and click “OK“.

See chapter: “Model -> Relative Element Size [%]“ before you generate a FE-mesh of your part.

The new mesh file is stored in the directory of the project- resp. STL file.

The name of the new FE-mesh-file has 4 characters without any extension.

16.1.4.2 NASTRAN

Single cavities, multiple cavities and family molds can be transferred into NASTRAN files.

(1) Click “File → Export → NASTRAN”.

⇒ The window "NASTRAN Export" appears on the screen.

(2) Select all the parts you like to transfer into a NASTRAN file. (*Selected parts are marked red.*)

(3) Click <Export>.

(4) Enter the name and click "Save".

16.1.4.3 STL

Single cavities, multiple cavities and family molds can be transferred into STL files (stereo lithography files).

(5) Click “STL”.

⇒ The window "STL Export" appears on the screen.

(6) Select all the parts you like to transfer into a STL file. (*Selected parts are red marked.*)

(7) Click <Text> or <Binary>.

(8) Enter the name and click "Save".

16.1.4.4 Compensation of Shrinkage and Warpage Results

(Separate auxiliary module of CADMOULD® 3D-F)

Part geometry can be transferred into different file formats with consideration of the arising deformation of the part during the cooling time. These files can be used for the numerical control and/or computerized numerical control treatment of the tool.

The following file formats can be exported:

- .rm1 – Cadmould 3D-F project file
- .stl – binary or text STL-file
- .cfe – Cadmould 3D-F FE-Mesh file
- .nas – NASTRAN file
- .igs – IGES file
- .txt – Catia file

Today a forecast on shrinkage arising at the part and the warpage of the part still is to be regarded with caution. However, CADMOULD® 3D-F offers to you a forecast of shrinkage and warpage of a part which was tested and worked in real life applications.

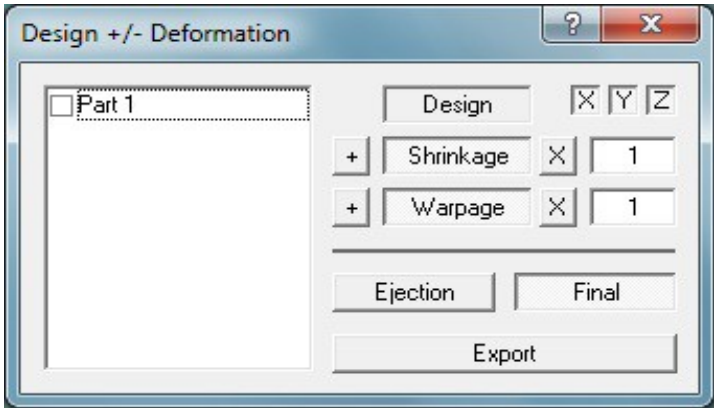
You can intervene manually in this compensation of a part geometry with shrinking and warpage results.

(1) Click on "Design + / - Deformation".

⇒ The window " Design + / - Deformation" appears on the screen.

(2) Select the part you want to transfer.

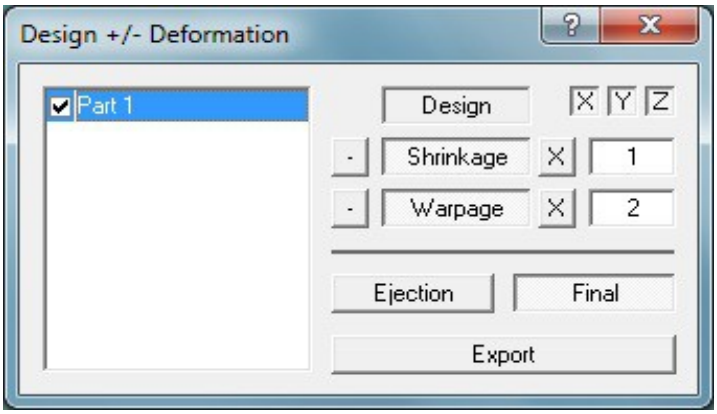
(3) Define the export type of the part geometry. The shrinkage and warpage values occur in positive directions, so for a compensation the negative signs are required: “[- Shrinkage], [- Warpage], i.e. the compensation must be applied in opposite direction to the shrinkage and/or warpage of the part, thus leaving the export larger than the given model.



By the buttons X, Y and Z you can define spatial directions for which the compensation is blocked, I.e. a pushed button means that a compensation for this direction will be performed.

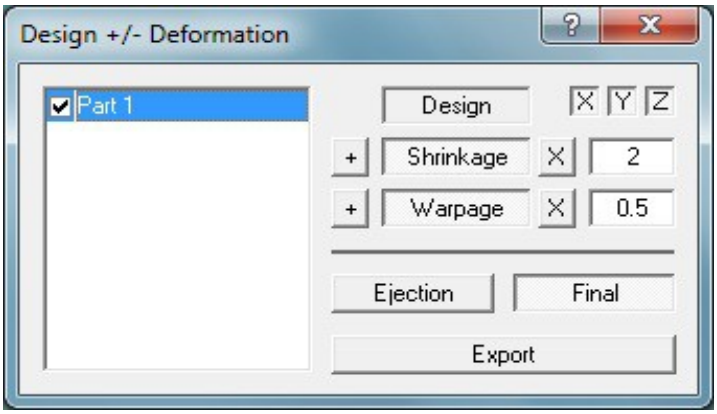
Example:

Cavity = Design + [– Shrinkage] x 1 +
[– Warpage] x 2 to twice overemphasize the warpage.



Example:

Part = Design + [Shrinkage] x 2 +
[Warpage] / 0,5 if the true shrinkage of the material is twice the simulated and the true warpage is only half the simulated (which is rather seldom for a well defined material and process).



(4) Click on <Export>.

The STL file format is a usual transfer file format.

⇒ The window "Save as" appears on the screen.

(5) Define a directory, a file name and click on <store>.

(*See* chapter: "S+W (Shrinkage and Warpage)";"Mesh Files")

16.1.4.5 Material Data

With the function “**Export - > Material Data**” material data can be copied from current projects. The material data are stored in a PLB-file (*Material*.plb) and can be imported into CADMOULD® 3D-F material data base or be led further.

Note: Material data is a component of the CADMOULD® 3D-F software. Consider please the copyright regulations.

16.1.4.6 Wall Temperatures

Wall temperatures can be transferred from current projects in TMPO-files (project_*identity-number*_TMP0) and be edited e.g. with the module „CADMESH“. Subsequently, the wall temperatures can be imported into the project.

(*See* chapter: "Wall Temperatures")

16.1.5 Convert

16.1.5.1 CAR -> Txt

CAR-files are result files. The result files can be converted into TXT-files by the function “Convert / CAR -> Txt”. This may be of interest if you want to interface results to other programs.

Convert the CAR-files by opening them. The TXT-files are automatically stored in the directory of the CAR-files, replacing the extension.

(1) Click on "CAR -> Txt".

⇒ The window "Open" appears on the screen.

(2) Select a type of result in the list field „Files of Type“.

- ***Cadmold***-Any-Result-Files
- Flow Front
- Snapshot Filling
- Air Traps + Weld Lines
- Diagrams
- Orientation
- Layer Temperatures for S+W Analysis
- Residual Stress
- Part Quality
- Snapshot Packing
- Shrinkage + Warpage - Cooling
- Shrinkage + Warpage (Ejection)
- Shrinkage + Warpage (Final)
- Snapshot Filling Temperature Profiles
- Snapshot Filling Velocity Profiles
- Snapshot Packing Temperature Profiles
- Snapshot Packing Velocity Profiles
- Residual Stress in Layers
- Wall Temperature Analysis
- Sensors

(3) Select a file.

(4) Click on "Open".

The opened file was converted into a text file and stored in the same directory as your STL-file.

16.1.5.2 RM1 -> CFE

This function makes possible to extract the FE-Mesh from a RM1/DES/SYM-file and save it into a CFE-file.

Choose the files you would like to convert in the dialogue an click on "Convert".

16.1.5.3 BMP/PNG -> AVI

With this function you can create an AVI-file from a sequence of pictures. Every picture is shown in the AVI for the same period.

(1) Click “BMP/PNG -> AVI”.

⇒ The file explorer appears on the screen.

(2) Select the sequence of pictures you want to convert.

(3) Click on "Convert".

⇒ The window "Save" appears on the screen.

(4) Choose a file name for the animation.

(5) Click on "Save".

16.1.6 Explore Project Directory

Opens the Windows Explorer in the project folder.

16.1.7 Auto Store

CADMOULD® 3D-F provides, if the function is activated (standard), automatically a project backup file. In the field “Minutes” time interval between 1 and 60 can be given for the actualization of the project backup file.

In case of incorrect abort of a project, the project can be repaired and stored by clicking <Restore Session> and entering a title and identity number of the project in the new appeared window.

16.1.8 Save Animation

You can save an animation of time based results.

Please see chapter: Store Animations

16.1.9 Save Image

The window "Save as" appears on the screen. You can store the current picture as a BMP-file (bitmap-file) or a PNG-file (Portable Networks Graphics-files).

Besides the possibility to save a well-known 2D-Picture, you can save a 3D-Picture. To this there are three 3-Dimensional image formats available.

You can find an introduction to the 3D-formats here:

Https://en.wikipedia.org/wiki/X3D

Http://en.wikipedia.org/wiki/Vrml

On the following web pages you can find programs to open these 3D-formats

http://www.web3d.org/x3d/wiki/index.php/Player_support_for_X3D_components

http://castle-engine.sourceforge.net/view3dscene.php

http://www.instantreality.org/downloads/

http://www.bitmanagement.com/de/download

Simcon dissociates itself hereby expressly from all designs and contents of all linked sites on this manual. simcon does not make these designs and contents its own.

16.1.10 Save Window

With this function you can take a screenshot of the opened windows. The window you select can be saved as PNG-file (Portable Network Graphic-File) or BMP-file (Bitmap-File).

Therefore point with the mouse pointer on the window you want to save.

⇒ The window “Save As” appears on the screen.

Chose a file name and file type before clicking “Save”

16.1.11 Print

In the menu "Print" you can define the printer characteristics and print your part.

For more information about the setting of the printing characteristics refer to your Windows and printer manuals or help function.

16.1.12 Print Option

The window "Print Option" appears on the screen.

In the field: "Image Size [%]" you can adjust the size of the print.

100% corresponds to DIN A4 sheet size.

50% correspond to DIN A5 sheet size.

In the field: "Resolution Reduction" you can set the factor, by which the print resolution is reduced.

16.1.13 Print Setup

The window "Print Setup" appears on the screen. This menu is used in the same way as Windows print setup.

For setting the printer characteristics you can refer to your Windows and printer manuals.

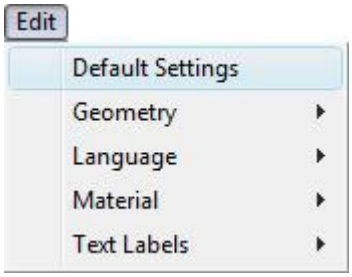
16.1.14 Recent File List

The 9 last opened files are in a list in the menu “File” for fast access.

16.1.15 Exit

Click “Exit” and “Yes” to finish CADMOULD® 3D-F.

16.2 Edit



Click "Edit".

⇒ The pull down menu "Edit" is activated.

In the pull down menu “Edit” you find instructions, which make changes at your part of mould and the project possible.

16.2.1 Default Settings

Restores the default settings of CADMOULD® 3D-F.

16.2.2 Geometry

16.2.2.1 General

16.2.2.1.1 Scale

STL - files are dimensionless, i.e. they contain only the numerical values of a certain dimension. CADMOULD® 3D-F always assigns the length "millimeter" to the numerical values of a STL-file.

In order to assign the appropriate size to the part, runner or heating/cooling system, you must multiply the numerical values of given case by a scaling factor.

You can select also the scaling factors freely, and adapt so the size of part geometry, the keeping at a moderate temperature system or the dead head system.

(*See* chapter: **STL**)

(1) Click on “Scale”.

⇒ The window “Scale” appears on the screen.

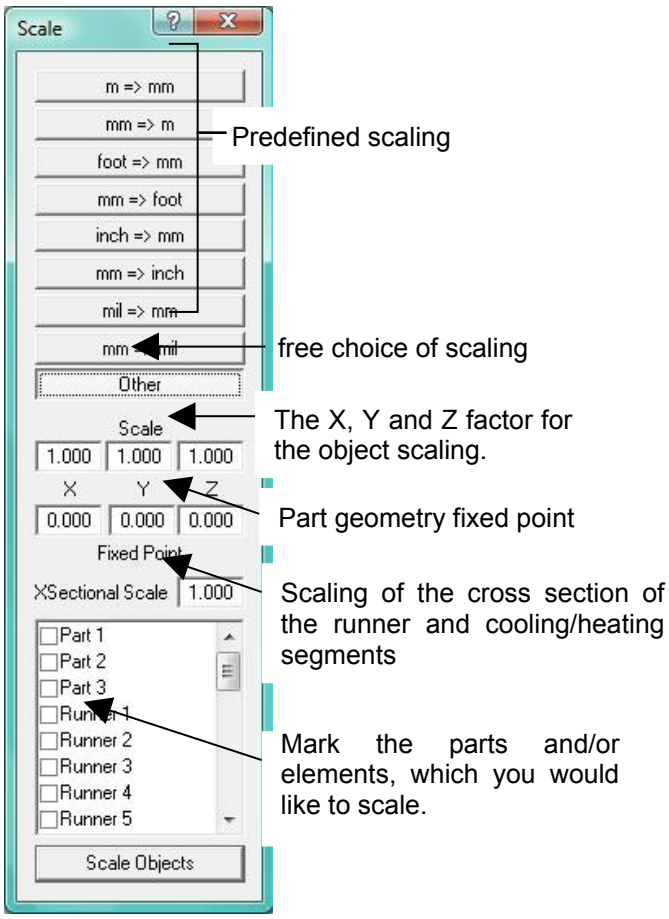
(*See* Point on ... function)

(2) Determine the scaling.

(3) Mark your part.

(4) Click on <Scale Objects>.

(*See* chapter: Load at the beginning)



16.2.2.1.2 Transform

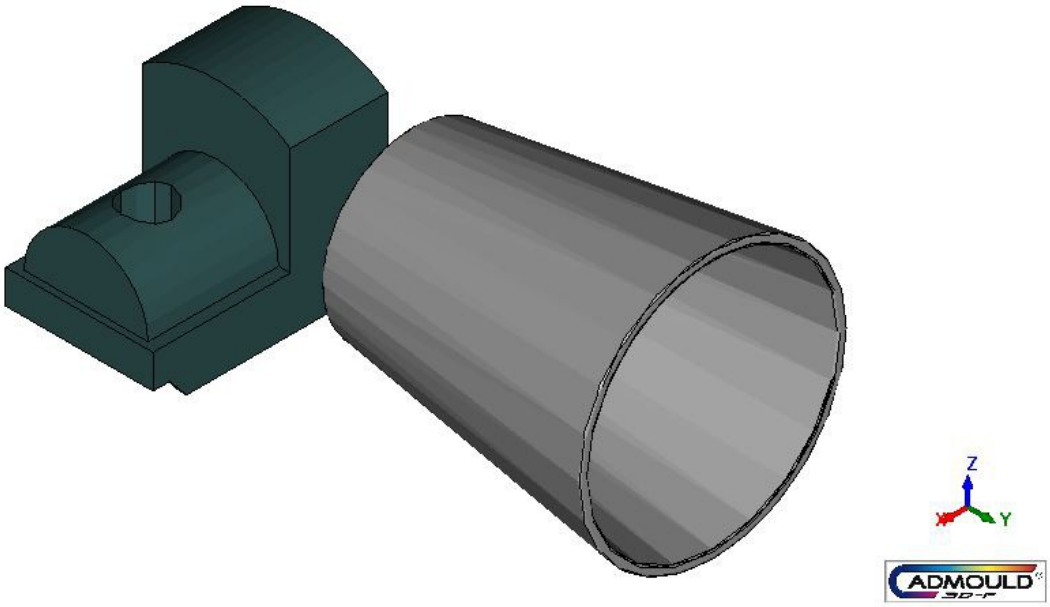
You can change the spatial position of a part by using the function “Transform”. The projected part on the screen always will be transformed into the following system of coordinates. Because of no changing the aspect of the part, it seams as if the surrounding adapts to your part.

The functions “Transform” and “Look vertically at...” offer a easy way to position particular parts in a mold.

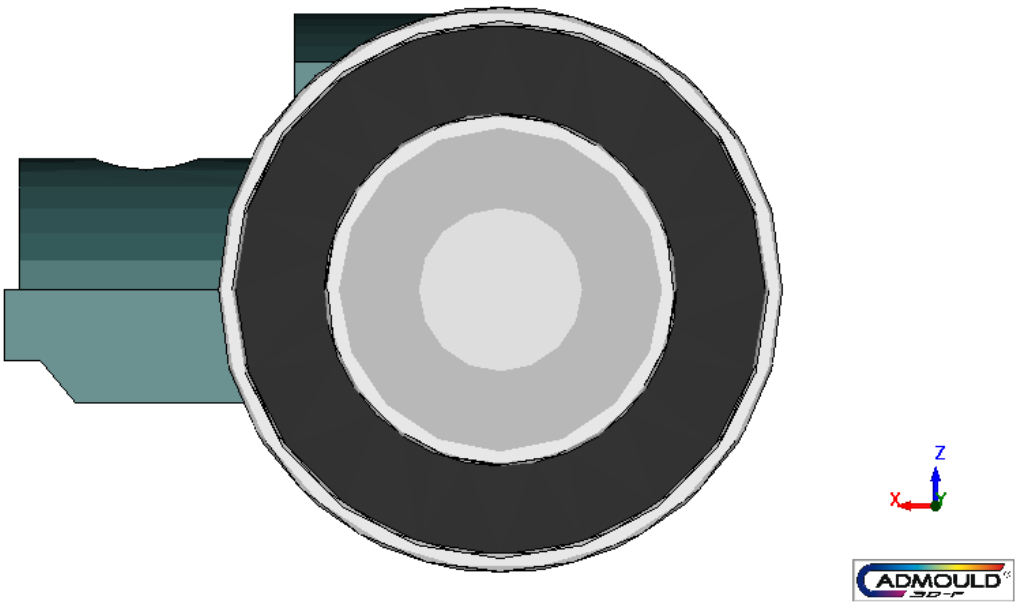
(See chapter: **“Look vertically at...”** and the following example: **Mold Position***)*

Example: **Mold Position**

1) Load two different parts.

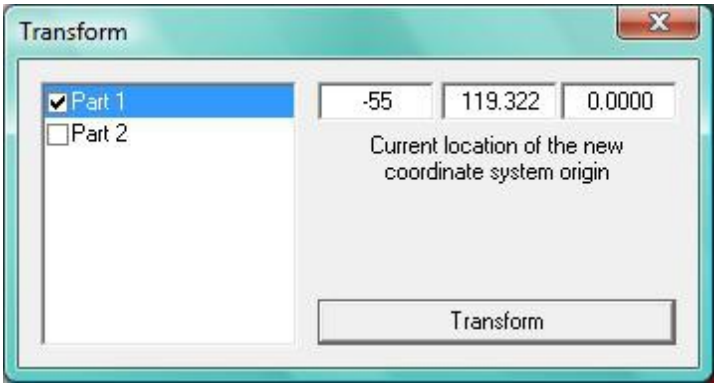


2) Define the aspect to the first part.



3) Click on “Transform”.

⇒ The window “Transform” appears.



4) Mark the part you want to transform. (The cup is part 1.)

5) Set the cursor in one of the three fields of the coordinates of the coordinate system origin.

6) Keep the <Ctrl> key pressed and click on your part where you want to put the coordinate system origin.

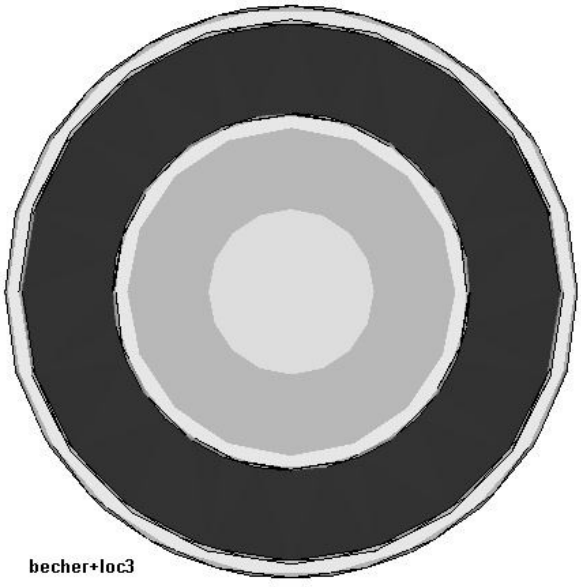
(*See* Point on ... function)

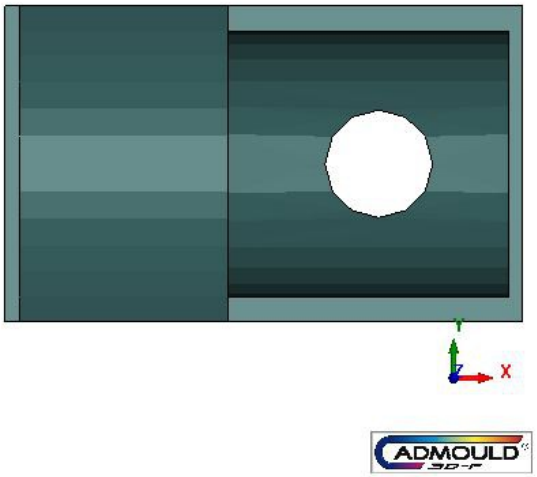
⇒ The coordinates appear in the input fields.

7) Click on <Transform>.

Note: All parts which were not marked keep their relative position to the coordinate system origin. They are shifted and turned together with the coordinate system origin.

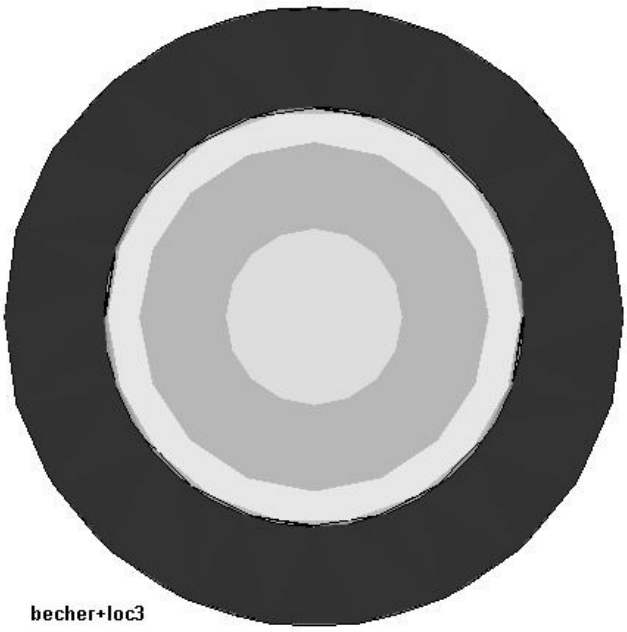
⇒ You get a new location of the transformed part.

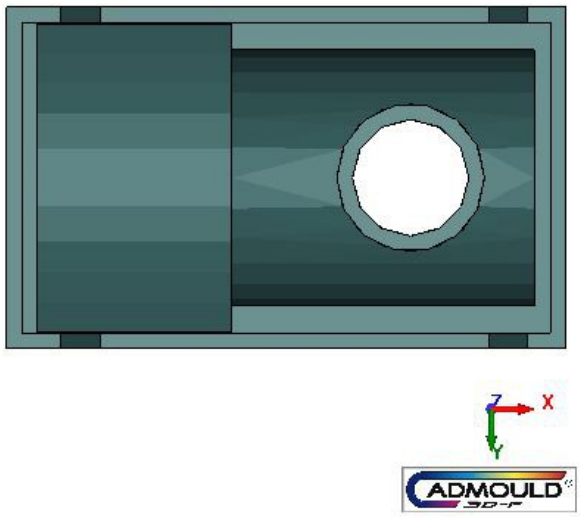




For orientation, compare the new situation of the part in the coordinate system with the new coordinate system origin’s position.

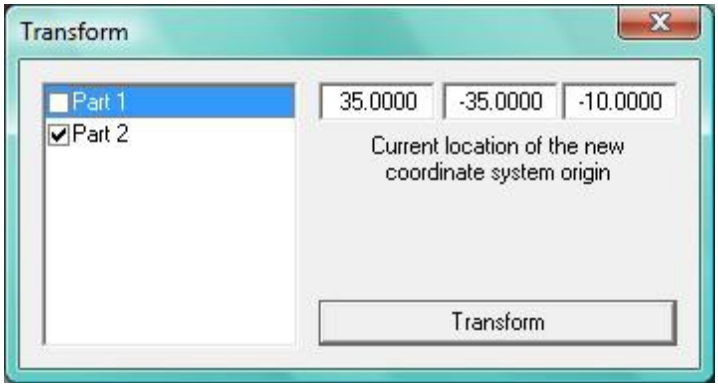
8) Define the aspect to the next part.





9) Click on “Transform”.

   ⇒ The window “Transform” appears on the screen.



10) Mark the part you want to transform. (Locomotive = part 2)

11) Set the cursor in one of the three fields of the coordinates of the coordinate system origin.

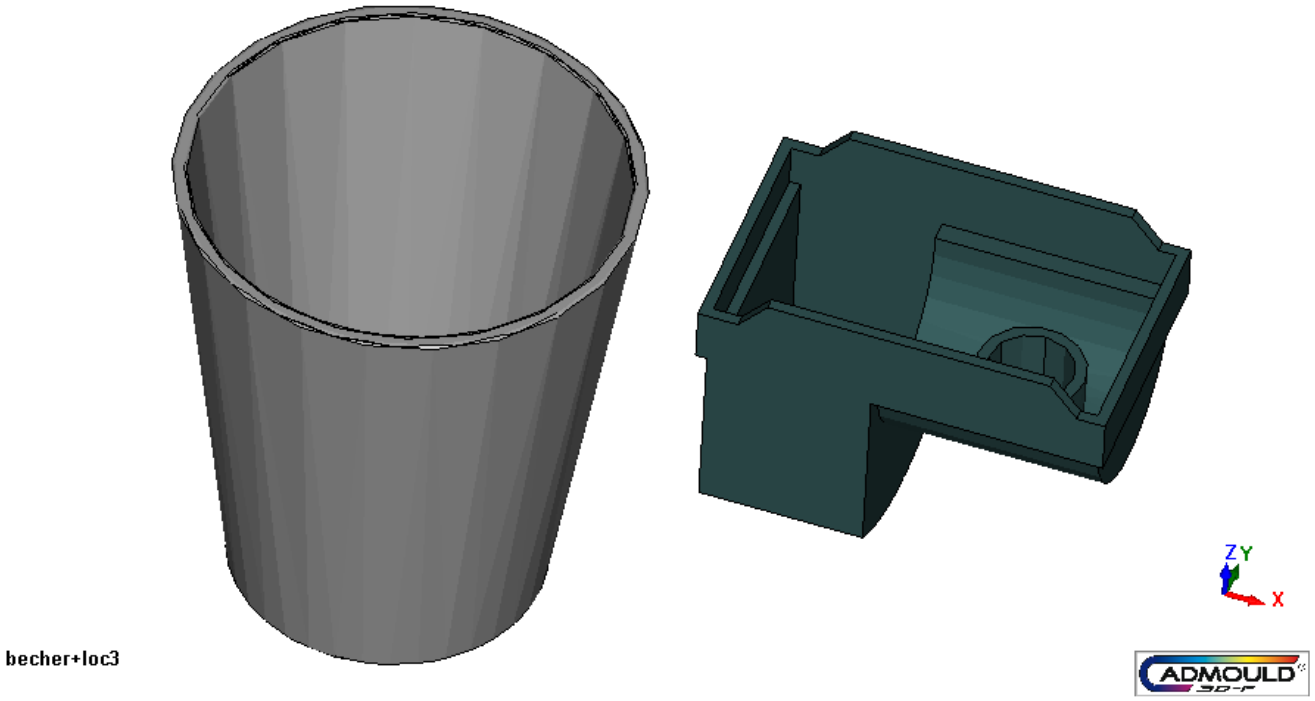
12) Keep the <Ctrl> key pressed and click on your part where you want to put the coordinates system origin or type in the exact position. (*See* Point on ... function)

   ⇒ The coordinates appear in the window.

13) Click on <Transform>.

Note: All parts which are not marked keep their relative position to the coordinate system origin. They are shifted and turned together with the coordinate system origin.

⇒ You get a new location of the transformed part.



For orientation, compare the new situation of the part in the coordinate system with the new coordinate system origin's position.

If you want to define the new coordinate system origin only, then you need to set the base view, mark all the parts in the window "Transform", define new coordinates of the origin and then click on < Transform >.

16.2.2.2 Groups

The instruction: "**Groups**" gives you the possibility to group part elements and to allocate group characteristics. Individual groups can be faded in and faded out. Part elements or part objects can be assigned to different groups. The element and/or object selection can be done by using selection tools.

(*Consider please* the chapter: "Heat Transfer Coefficient")

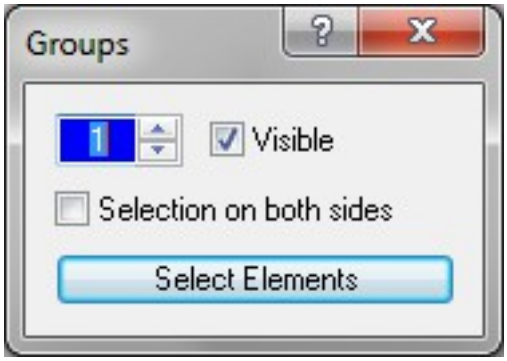
The element and/or object grouping can be accomplished before a mesh preparation, i.e. also at a STL file to allow group specific element sizes.

Grouping elements:

1) Click on „Group".
   ⇒ The window "Group" appears at the screen.
2) Select a group by clicking the arrows up or downward.
3) Click on <select objects>. (*The window: "Result Selection" should be closed.*)

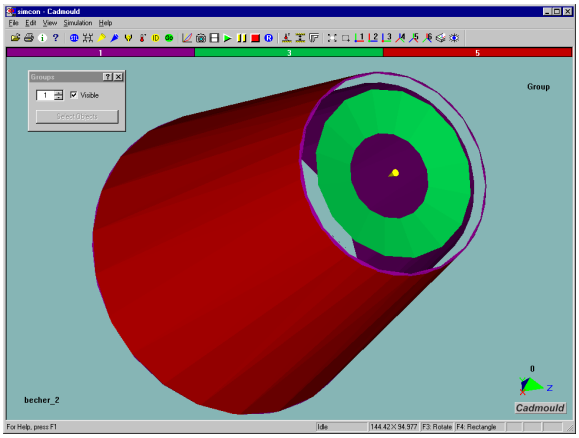
⇒ The window "Groups" disappear.

4) Select the elements, you like to assign to the selected group.
   (*See* Table: Selection Tools)



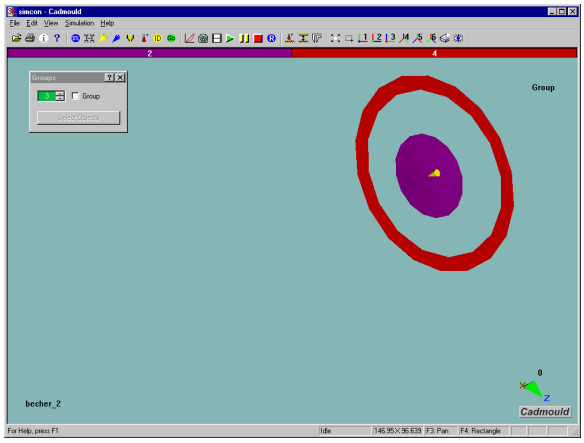
5) Confirm the regrouping in the following window.

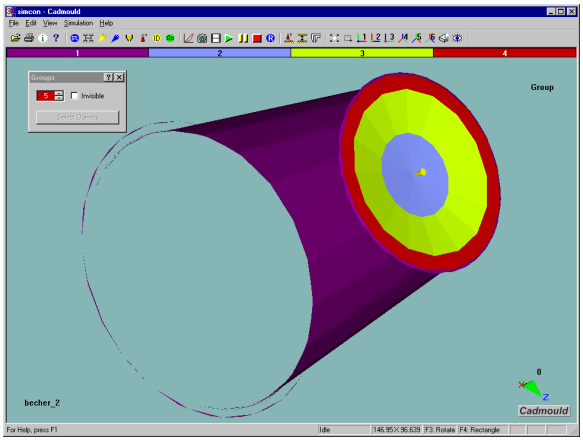
Example: Groups fade in and groups fade out



All faded in groups are provided with a checkmark.

The checkmark is missing at faded out groups.





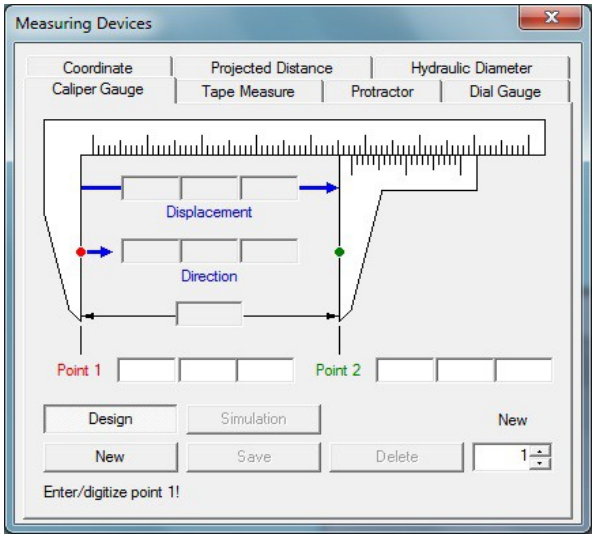
16.2.2.3 Heating/Cooling Channel

This function contains the commands Runner segment, 3D Heating/Cooling Channel and Heating/Cooling Circuits

16.2.2.4 Measuring Devices

This function opens a window where you find the following tools, to measure the part.

Caliper Gauge



(1) Load a part.

(2) Click on "Measuring Devices".

⇒ The window "Measuring Devices" appears at the screen.

(3) Stay on the register "Caliper Gauge".

(4) Next two points must be defined.

The next step is additionally shown in the lower left corner.

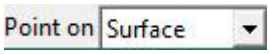
The "point on..." function helps to select corner points exactly.

As result of measurement you receive:

- the distances in X -, Y and Z-direction
- the direction in form of a unit vector
- the direct distance of the given points

- the difference (absolutely, in per cent) between the designed and the simulated part

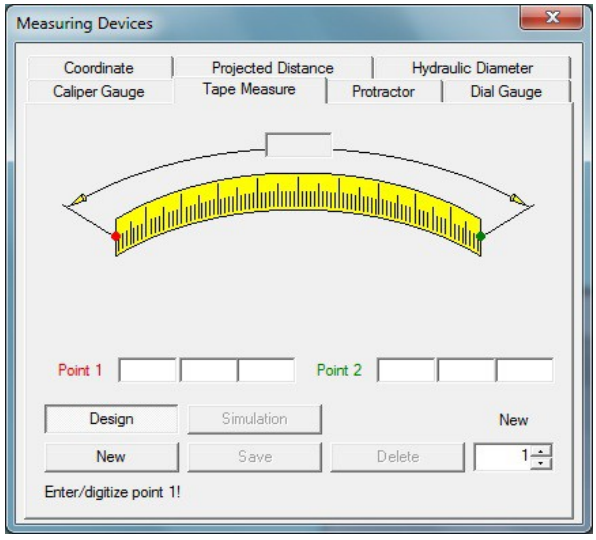
Point on ... function



The “Point on...” function is from now on located in the status bar of CADMOULD® 3D-F. This function helps you to select a point on the surface of the part. The pull-down menu is only visible, if a selection of a point is necessary.

- Activate the option „Surface“, keep the button <Ctrl> pressed and click on a surface, in order to select one point on the surface.
- Activate the option „Edge“, keep the button <Ctrl> pressed and click near to an edge, in order to select one point on an edge.
- Activate the option „Corner“, keep the button <Ctrl> pressed and click near to a corner, in order to select a corner on the surface.
- Activate the option „Node“, keep the button <Ctrl> pressed and click near to a node, in order to select one node on the surface.
- Activate the option „Group Border“, keep the button <Ctrl> pressed and click near to a group border, in order to select one node on the group border.

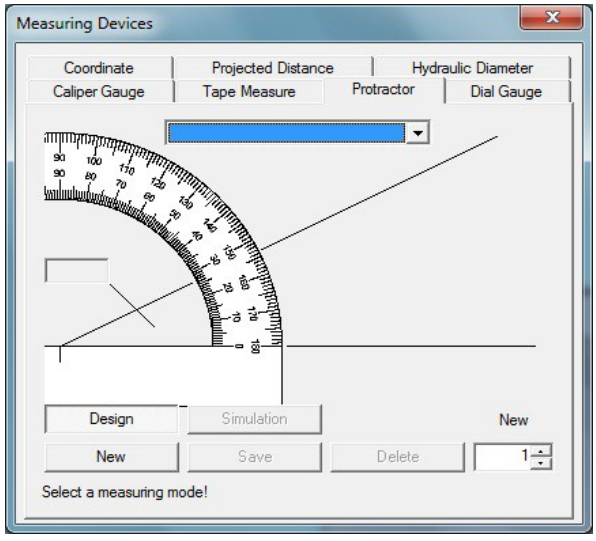
Tape Measure



(1) Load a part.
(2) Click on "Measuring Devices".
⇒ The window "Measuring Devices" appears at the screen.
(3) Switch to the register "Tape Measure"!
You must now define the two points, of which you want to measure the distance.
The results you get are the following:

- the shortest geodetic distance between two points
- the difference (absolute, percental) between the designed and the simulated part

Protractor



(1) Load a part.
(2) Click on "Measuring Devices".
⇒ The window "Measuring Devices" appears at the screen.
(3) Switch to the register "Protractor".
(4) Choose one measurement method from the list. (for example the "3 Points" method)

(5) Next three points must be defined.

The next step is additionally shown in the lower left corner.

The "point on..." function helps to select corner points exactly.

(See Point on ... function)

As result of measurement you receive:

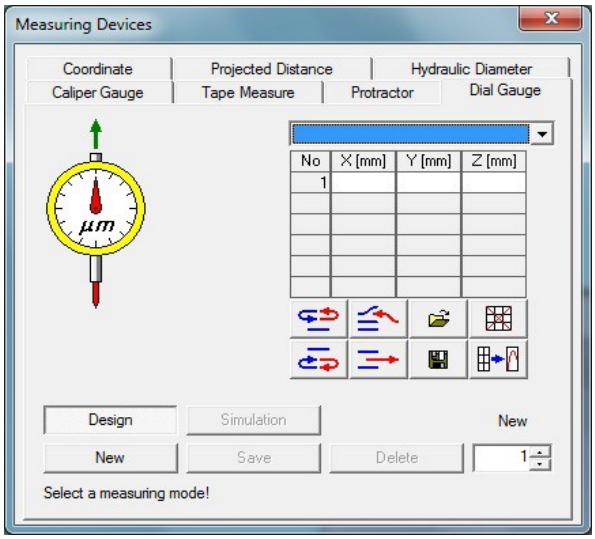
- the angle defined by three/four points
- the angle at an edge
- the angle between two edges/surfaces
- the angle between an edge and a surface
- the difference (absolutely, in per cent) between the designed and the simulated part

Dial Gauge

(1) Load a part.

(2) Click on "Measuring Devices".

⇒ The window "Measuring Devices" appears at the screen.



(3) Switch to the register "Dial Gauge".

(4) Choose one measurement method from the list. (for example the "Roundness" method)

(5) Next the axis vector must be entered/digitized.

(6) Now you can choose a point on an edge/surface of the part.

The next step is additionally shown in the lower left corner.

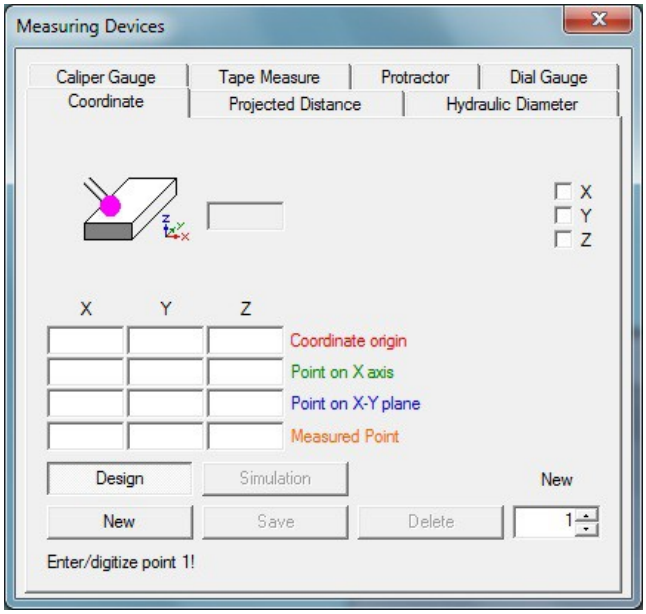
The "point on..." function helps to select corner points exactly.

(See Point on ... function)

As result of measurement you receive:

- the straightness of an edge
- the flatness of a surface
- or the roundness, respectively shown as a diagram
- the difference (absolutely, in per cent) between the designed and the simulated part

Coordinate



1) Load a part.
2) Click on "Measuring Devices".

⇒ The window "Measuring Devices" appears at the screen.

3) Switch to the register "Coordinate".

You have two possibilities for the alignment procedure: **<3-Click Transformation>** and **<3-2-1 Method>**.

Exemplification of **<3-Click-Transformation>**

Enter the origin of coordinates in the first line of the matrix. This point must be on the surface of your part. Click – while holding the Ctrl-Key on the part.

In the next step, you define the direction of your X-axis. Click while holding the Ctrl-Key on the part. The vector from the first to the second point is the X-axis of your coordinate system.

Now, you will define the position of the X-Y-plane. The vectors 1-2 and 1-3 define the X-Y-plane, the z-axis is perpendicular to it.

In the last row you enter the measuring point.

You can select one or more components (X,Y,Z) to save.

The saved measures are the X/Y/Z component of the position vector from the origin of your defined coordinate system to the measuring point. For each component one measure is saved.

Exemplification of **<3-2-1 Method>**

The points A1, A2 and A3 define the X-Y-plane. Perpendicular to it is the z-axis. In the next step, you enter the points B1 and B2 to construct the direction of the X- and Y-axis. Y=0 is on the projection of the vector B1B2 to the X-Y-plane.

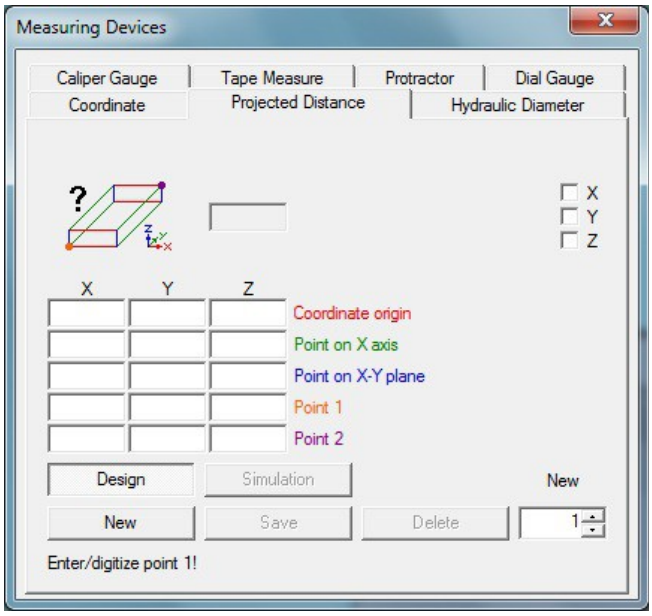
C1 defines finally the X-Axis.

Your reference coordinate system is defined.

1. Set your measuring point onto the part (Click, while holding the Ctrl-key, onto the part).
2. Select one or more components (X/Y/Z) to save.

The saved measures are the X/Y/Z component of the position vector from the origin of your defined coordinate system to the measuring point. For each component one measure is saved.

Projected Distance



1) Load a part.
2) Click on "Measuring Devices".

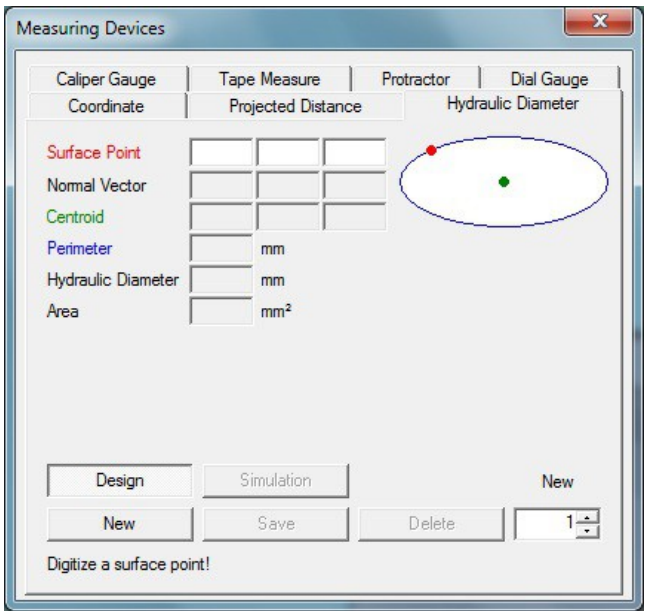
⇒ The window "Measuring Devices" appears at the screen.

3) Switch to the register "Projected Distance".

You have two possibilities for the alignment procedure: **<3-Click Transformation>** and **<3-2-1 Method>.** The Exemplifications about the alignments can be gathered under the section “Coordinate”.

4) Choose now one or more components (X, Y, Z), which will be saved as gauge by clicking “Save”. Additionally the according projected planar resp. Spacial distance is save, if two or more directions are chosen.

Hydraulic Diameter



1) Load a part.
2) Click on "Measuring Devices".

⇒ The window "Measuring Devices" appears at the screen.

3) Switch to the register "Hydraulic Diameter".
4) Digitize a point on the surface. You will get the normal vector, the center of gravity, the circumference, the hydraulic diameter and the surface area of the collected cross section.

These function can be helpful to determine the hydraulic diameter of runner segments which are available as STL file.

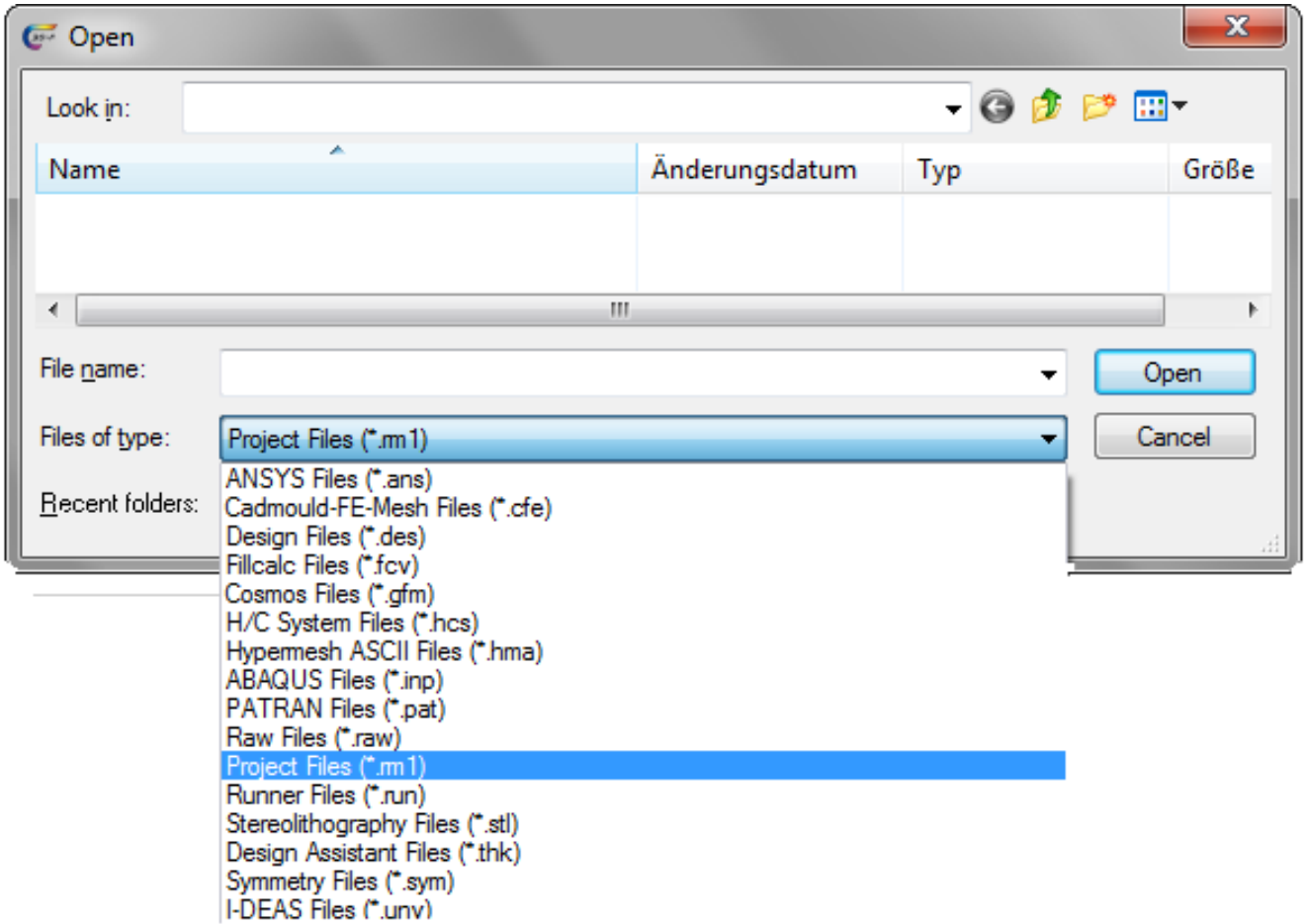
16.2.2.5 Mould → Mounting Position

Usually parts are delivered as STL files, where they are in mounting position. For the design of multiple-cavity moulds and a proper simulation of clamping forces the parts are repositioned by the user. Then in the final stage of a project geometrical data (gating) and modifications (flow leaders) have to go back to the part designers, where they are expected based on the mounting position again. With this dialog the mounting position of parts can be restored after an unknown sequence of rearrangements of parts. To accomplish this a triangle of characteristic points has to be digitized in both the mould position and the mounting position. For precision the triangles should be as large as possible and be made of points that are not relocated by the mesh pre-

paration. We recommend to digitize distant sharp corners with the digitizing mode “Corner”. Provided both triangles are equal in size (preserve scale) and orientation (avoid mirroring) CADMOULD® 3D-F calculates the transformation required to put parts back into place. The accompanied file selection dialog takes care of loading a broad variety files, which may serve to define both positions.

Please note, that the result-files 110 (Unwarp), 120 (Ejection) and 121 (processing) will be transformed as well.

Dependent on your license, there is a large selection of file formats which are able to save the definition of Mould- and Mounting Position.



16.2.2.6 Parts

16.2.2.6.1 DeskArtes

The DeskArtes programs "3Data Expert" and "SimExpert" (starting with version 10.0) can repair parts selected in CADMOULD® 3D-F and then return the result via "File - Update in Cadmould" immediately back to CADMOULD® 3D-F, where the defective part is replaced automatically. If in CADMOULD® 3D-F more than a one part is present, a dialog "DeskArtes" appears, similar to the dialog "Delete" to

select a part for repair. If only a single part is present, the DeskArtes program will appear with the

part loaded immediately.

1. Import a STL file into CADMOULD® 3D-F.

2. Check, if the STL-Part is correct. Incorrect areas are represented in different colors as the rest of the part.
3. If your part is not correct, click **<Edit – Geometry – Parts – DeskArtes>**.
   → The program DeskArtes starts with the incorrect part.
4. Repair the STL in DeskArtes.
5. In DeskArtes, click **<File – Update in Cadmould>**
6. The defectve part is replaced auomatically.

To use this function one of the programs “3Data Expert” or “SimExpert” from version 10.0 must be installed on your PC.

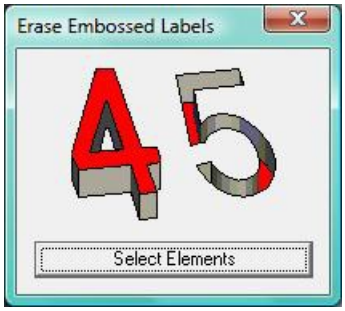
16.2.2.6.2 Element Orientation

This function shows the orientation of the elements and repairs it (only in CADMOULD® 3D-F). Select the elements with the well known tools, which can be chosen by pressing the F4 button. Faulty elements (“holes” in the geometry) can be repaired easily by switching the blue elements to red.

16.2.2.6.3 Erase Embossed Labels

Thus the simulation of your part is not extended unnecessarily, embossed labels should be removed from your part. Embossed labels extend the phase of mesh preparation and change the CFE – mesh quantitatively. The consequence is a longer duration of the simulation.

(1) Click on “Erase Embossed Labels”.
   ⇒ The window “Erase Embossed Labels” appears on the screen.
(2) Click on <Select Elements>.
The selected elements are marked red.
The embossed labels, which can be deleted, are marked yellow.

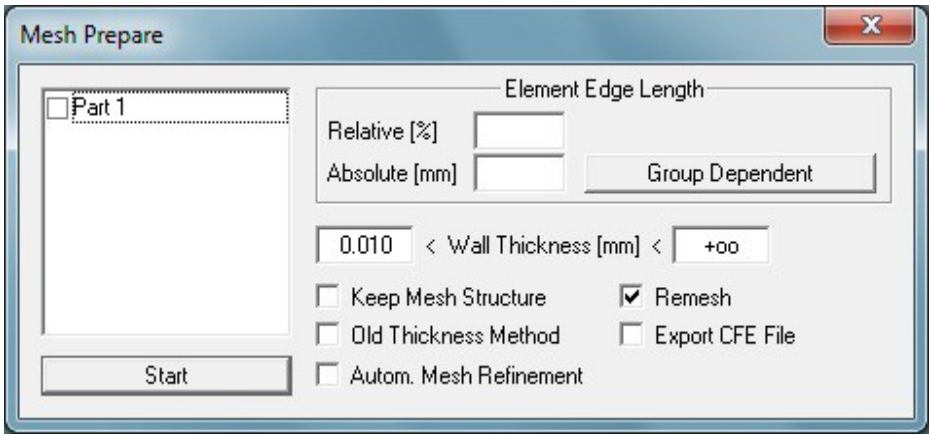


16.2.2.6.4 Mesh Prepare

This function allows to prepare the mesh of one or more molds without entering the usual inputs (distribution system, material, process, identifier). It´s possible to make different settings for every mould.

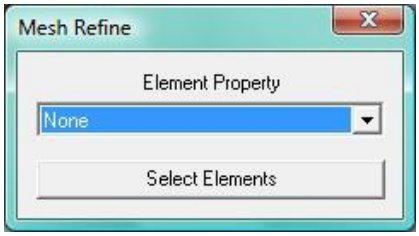
1) Load your geometry.
2) Click "Mesh Prepare".

⇒ The window "Mesh Prepare" appears at the screen.



3) Select the parts you want to prepare. They appear blue in the list and red at the display.
4) Now choose the options for the mesh preparation. You can define “Group Dependent” lengths if you have defined groups before. This could be usefully, if your part contains very thin areas which should be meshed denser, e.g. living hinges. For this click on “Group Dependent” and enter the lengths in the table. Always additionally specify a value in the main window too.
5) Click <Start> to start the mesh preparation.
6) After the preparation, the prepared meshes will be exported with the file names "Part001.cfe", "Part002.cfe" and so on, if this option is selected.

16.2.2.6.5 Mesh Refine



In order to get better result, you can refine the FE-mesh manually.
(*Consider please* the chapter: Automatic Mesh Refinement)

1) Click "Mesh Refinement".

⇒ The window "Mesh Refinement" appears at the screen.

Before you start with the mesh refinement you can indicate the mesh characteristics.

- Thickness [mm]
- Group
- Wall Temperature Filling [°C]
- Wall Temperature Cooling [°C]
- Heat Transfer Coefficient [W/(m²K)]
- Element Orientation

Click on <select objects>. (*The window: “Result Selection” should be closed.*)

⇒ The window “Mesh Refinement” disappear.

(*See* Table: Selection Tools)

2) Select the FE-mesh elements, you like to refine.

(*See* chapter: Element Edges)

3) Confirm the mesh refinement in the following window.

16.2.2.6.6 Thickness

In order to be able to change the wall thickness, you need a display set to **more** than 65536 colors ((= High Color = 16 bit), i.e. True Color (= 24 bit or 32 bit)). This requires a graphic adapter/monitor combination that allows running this setting.

If your PC fits these requirements, you can change the wall thickness in the entire part, as well as in sub-areas of the part.

Changing the wall thickness

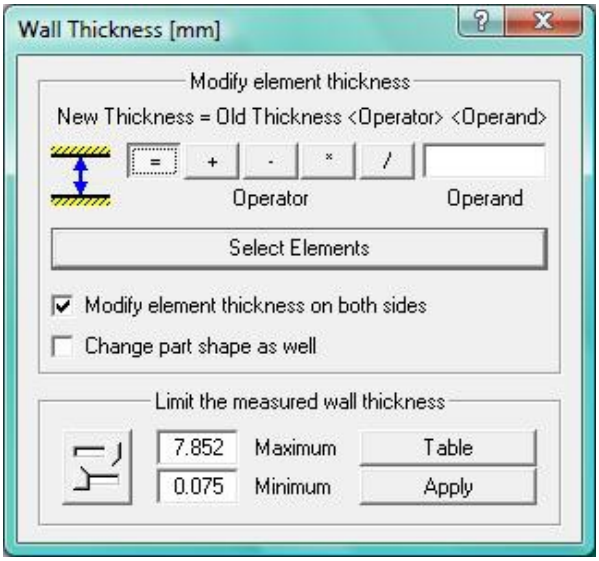
(8) Adjust the correct color depth, screen resolution and the sync frequency. (see chapter: "Thickness - Adjust the system settings")

(9) Load your part for which you must have a run a simulation or at least the mesh preparation phase.

(10) Click "Thickness".

⇒ The window "Wall Thickness [mm]" appears on the screen.

(11) Check local thickness values by pressing the <Ctrl> key and clicking onto the part. The thickness will be displayed on the status line at the bottom.



Follow the instructions in the window:

(12) Chose an operator (=, +, -, x, /). This will by applied to the area selected in the next steps.

(13) Give a value (operand) fitting your plan.

(14) Click "Select Elements". (*The window: "Result Selection" should be closed.*)

⇒ The window: "Wall Thickness [mm]" disappears.

(15) Select Elements using the Selection Tools listed in the following table.

Table: **Selection Tools**

Tool	Description
RECTANGLE	All elements completely enclosed by the rectangle
CIRCLE	All elements completely enclosed by the circle
POLYGON	All elements completely enclosed by the polygon
PLANE SURFACE	All elements on a surface
INFINITE PLANE	All elements on a plane
BOUNDED SURFACE	Surface definition based on the selected element situation to his neighbor elements. (*An angle have to be defined*)
DISTANCE	All elements , with a shorter distance along the surface to the selected element, than a selected value.
PART	Several parts

ALL	All elements
GROUP	All elements being member of the selected group
OBJECT GROUP	All elements of an object group
SINGLE	Single element (*Selection end => right mouse button*)

Switching between the individual tools:

F4 or Shift + F4 by an active CADMOULD® 3D-F operating surface.

Example: Selection tool **RECTANGLE**

(16)Drag opens a rectangle around the area to be modified.

⇒ By an active option *"☑ **Change part shape as well**"* the CFE – mesh is actually changed on one or both sides *(option: "☑ **Modify elements on both sides**")* by the given arithmetic operation.

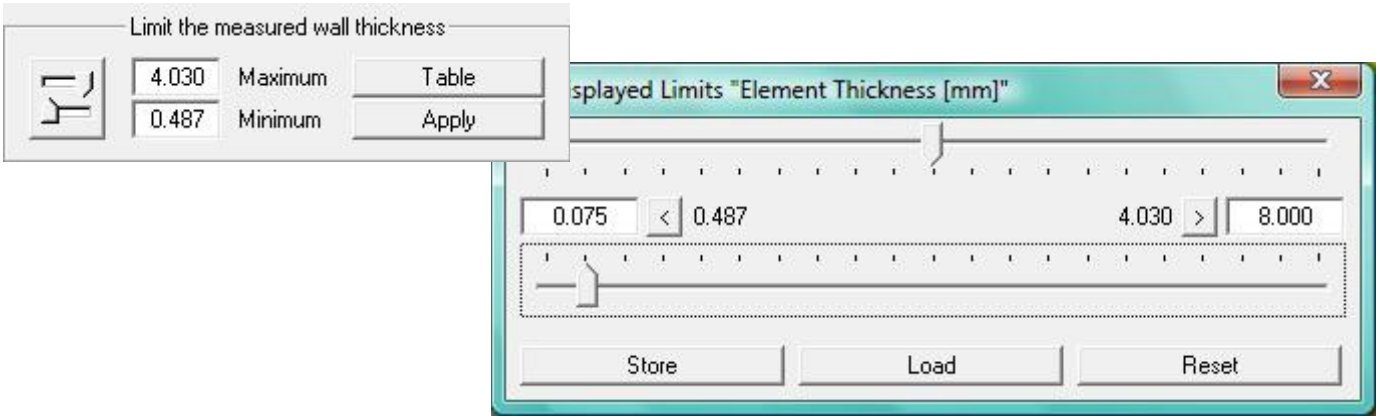
The CFE - mesh and concomitantly the volume of the shaped part change.

(17)Repeat the operation to change all elements to be modified.

(18)Run a new simulation.

Limit the measured wall thickness

The measured wall thickness can be limited by the function "Displayed Limits" and with the key "Apply" the limited wall thickness interval can be forced for the next simulation.



All too thin triangle elements and wall to wall bounds are set on the interval lower limit. The too thick triangle elements are set on the upper limit, while the too thick wall to wall bounds are removed.

Thickness - Table

The button "Table" in the dialog "Wall Thickness [mm]" opens a thickness sorted element table. By clicking the column-head, the table will be sorted after this column. Clicking one of the elements in the table will show a blue cone on the element in the geometry.

Note: This cone can partial not be visible. To find the cone, zoom out of the part, until you see it, because the cone, different of the part doesn't changes its size.

Clicking an element on the part surface will move the table to this element in the list. You can edit the thickness in this table.
It is very easy to find and to correct errors in the mesh. This can be a useful help, if you didn't set the limits of the thickness in Simulation → Options → Model. This option is not available in CADMOULD® 3D-F RAPID

Thickness - Adjust the system settings

Be sure to have the rights to modify the settings.
(1) Click "Start" and go to "Settings"
(2) Click "Control Panel".
⇒ The window "Control Panel" appears on the screen.
(3) Activate "Display" with double-click.
(4) Click "Settings".
(5) Adjust the system settings to "True Color". Mind that changing the color depth may affect the display resolution and the sync frequency, changing them to unacceptable low values.

16.2.2.7 Runner

This function contains the command "Runner segments".

16.2.2.8 Symmetry

16.2.2.8.1 Build Symmetrical Cavity

As long as this dialogue is open you can build a symmetrical cavity from an existing arrangement of runner segments and partial parts using the functions "Mirror" and "Rotate". Different from the normal usage of these functions no new objects are created, but transformations which create mirrored resp. rotated virtual objects during the simulation. These virtual objects are visible as well. However, they cannot be selected without the original object. And they do not create simulation effort of their own, because they are filled identical to the original objects.
This dramatically reduces the simulation duration compared to the conventional simulation, without any loss in quality and completeness of the results. Push the "OK" button if you are satisfied with your work. To restart from scratch dismiss the dialogue with the "Cancel" button and reopen it.

16.2.2.8.2 Reconstruct w/o Symmetries

This function is the counterpart of the automatic "Symmetry Check" and the manual "Edit → Geometry → Symmetry → Build symmetrical cavity". Undoes symmetry utilization and rebuilds the cavity conventionally. This is required for the wall temperature analysis of symmetrical cavities, because the heating/cooling system result usually is not symmetrical.

16.2.3 Language

The language of the dialogs can be changed if necessary.

Click on "Edit" and select the appropriate language.

16.2.4 Material

16.2.4.1 Class

Allows you to choose the class of material. (Thermoplastic or Elastomer)

16.2.5 Text Labels

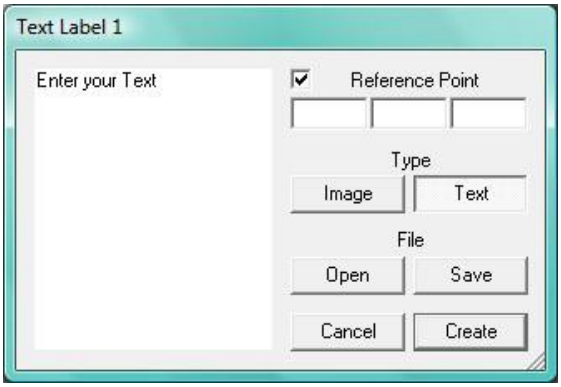
16.2.5.1 New

Project-related pictures and texts can be inserted into the CADMOULD® 3D-F surface.

Click on “Text Labels / New”.

⇒ The window “Text Label” appears on the screen.

♦ Enter your text:



You can enter a text directly into the text window. This text can be stored afterwards in the RTF file format.

Click on <Save>, in order to store the text. (i.e. *RTF files can be edit with Microsoft Word*.)

You can load RTF file. Click on <Open>, to load a RTF file. Pictures are not loaded.

Points of reference can be selected for each text label by Ctrl + left mouse button on the part. (*See* Point on ... function)

You can determine the size of the text label by pulling the right, lower corner of the window "Text Label".

Click on <Create>.

After you provided a label, the label can be:

edit by clicking => **Left Mouse Button**

positioned new by => **Middle Mouse Button**

removed by clicking => **Right Mouse Button**

(*See* chapter: Colors)

♦ Enter a picture:

BMP files (bit-map files)

PNG files (portable - networks - Graphics - files)

Points of reference can be selected for each picture label by Ctrl + left mouse button on the part.

(*See* chapter: Select Function)

(*See* Point on ... function)

(1) Click on <Image>.

(2) Click on <Open>.

⇒ The window “Open” appears on the screen.

(3) Mark an image file and click on <Open>.

⇒ The image appears in the window “Text Label”. You can determine the size of the image label by pulling the right, lower corner of the window "Text Label".

(4) Click the right mouse button on the image, in order to position it.

Position your Image, by selecting one of the options from the context menu.

(5) Click on <Create>.

After you provided an label, the label can be:

edit by clicking => **Left Mouse Button**

positioned new by => **Middle Mouse Button**

removed by clicking => **Right Mouse Button**

(*See* chapter: Colors)

16.2.5.2 Visible

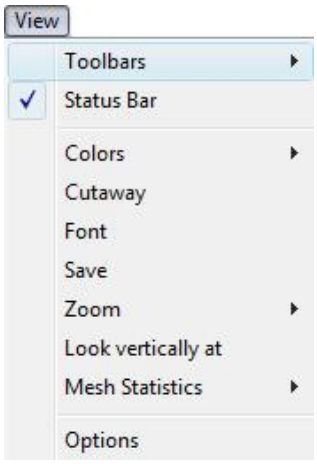
If the current project contains “Text Labels”, they can be made visible and/or be faded out by clicking on visible.

16.2.6 Wall Temperature

You have the possibility to assign wall temperatures to your part or to view the wall temperature distribution if you have carried out a wall temperature analysis.

Please see the chapter: Assigned Wall Temperatures

16.3 View



Click "View ".

⇒ The pull down menu "View" is activated.

In the pull down menu "View" you find instructions for customizing the user interface and changing the look of the mould part.

16.3.1 Toolbars

16.3.1.1 Visibility

The checkmark (√) shows the visibility of the “Toolbar“.

♦ Click “Toolbar“, in order to show or hide it.

There are the Toolbars “Standard”, “Edit” and dependent on the license also the bar “Wall Temperature” available.

16.3.1.1.1 Recently Used Functions

If you activate the toolbar “Recently used functions” there is a toolbar to which every menu entry you use is added, if there is a button for it. Entries already in default bars will not be added.

16.3.1.2 Configuration of Toolbars

You can now create your own button bars in Cadmould 3D-F to alleviate your working. In the following you get an overview of the functions which you can use to create your button bars.
These functions are also available in the context menu (right mouse button) of your own created button bars.

Customize Toolbar

This function helps you to customize your own toolbars, if you have already created one.
You can move the buttons with drag and drop.

New Toolbar

Create an initially empty user-defined toolbar:
Select "View - Toolbars - New Toolbar" from the pulldown menu.

Menu => Toolbar

With this function you can create a new toolbar from a menu.
Select "View - Toolbars - Menu => Toolbar" from the pulldown menu, select an item from the pulldown menu, enter a toolbar name. A new toolbar appears, containing all the items of the pulldown menu.

Rename Toolbar

You can rename your own toolbars.

Delete Toolbar

You can delete your own toolbars.

Add Button

With this function you can add a function from a pulldown menu to the end of your own toolbar. For this choose first this function from the menu and then the one you want to add.

Add Separator

With this function you can add a separator at the end of your own toolbar.

Insert Button

With this function you can add a function from a pulldown menu to a likely position in your own toolbar.
Therefore choose first this function from the menu, click than on the button to the right of the position where you want to add the button and at last choose the function you want to add from the menu.

Insert Separator

With this function you can add a separator to a likely position in your own toolbar.
Therefore choose first this function from the menu and click than on the button at the right hand side of the position where you want to add the separator.

Delete Button/Separator

With this function you can delete a button or a separator by clicking it.

Toolbar Help

This function opens a window with help for the described functions.

16.3.2 Status Bar

The checkmark (√)shows the visibility of the “Status Bar“.

- Click “Status Bar“, in order to show or hide it.

16.3.3 Colours

Define the colours of the objects “Air Trap”, “Background”, “Cutting Plane”, "Element Edges", “Flow Direction”, “Flow Path”, “Gate”, “H/C Circuit”, “Measuring Point”, “Part”, “Results”, "Selected Objects", "Sensor", “Text Label” and “Weld Lines” in the same manner as described below in the example “Air Trap”.

16.3.3.1 Air Trap

(1) Go to "Colours".

(2) Wait for a further menu to appear on the screen.

(3) Click “Air Trap“.

⇒ The window "Air-Trap Color" appears on the screen.

(4) Adjust the three sliders,

or

enter RGB components in the range 0 – 255 in the respective input fields,

or

click a color in the table.

(5) Click “OK” to activate the selected color.

16.3.3.2 Results

16.3.3.2.1 Default (color scale: 0 - 255)

(1) Go to “Colors” ⇒ “Results“.

⇒ A further menu appears on the screen.

(2) Click "Default".

16.3.3.2.2 simcon (color scale: 0 - 14)

(1) Go to “Colors“ ⇒ “Results“.

⇒ A further menu appears on the screen.

(2) Click "SIMCON".

16.3.3.2.3 Gray (black-and-white)

(1) Go to “Colors” ⇒ “Results”.

⇒ A further menu appears on the screen.

(2) Click "Gray".

16.3.3.2.4 Traffic Light

(Applicable for some quality results.)

(1) Go to “Colors” ⇒ “Results”.

⇒ A further menu appears on the screen.

(2) Click "Traffic Light".

16.3.3.3 Background

The option "Image ➔ White ☐" sets the background to white on images, independently from the screen background color. The current setting of the option is stored permanently in the registry. The default is not to use a separate white image background. The intention behind the option is to save ink when printing images, as the color white is left blank on printouts.

16.3.3.4 Parts

A color can be assign to each shaped part. Preset is gray.

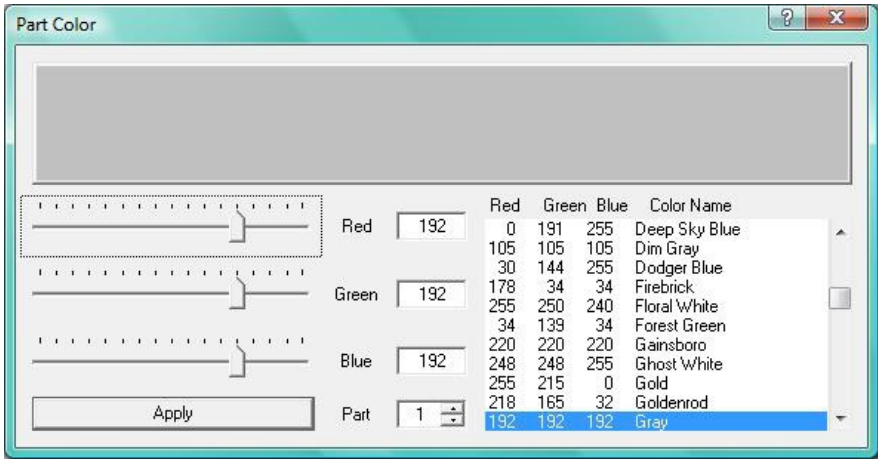
1) Click on “Part“.

⇒ The window "Part Color" appears on the screen.

2) Adjust the three sliders,

or

enter RGB components in the range 0 – 255 in the respective input fields,



or

click a color in the table.

3) Determine the shaped part, to which it is to be assigned.
4) Click <**Apply**> to activate the selected color.

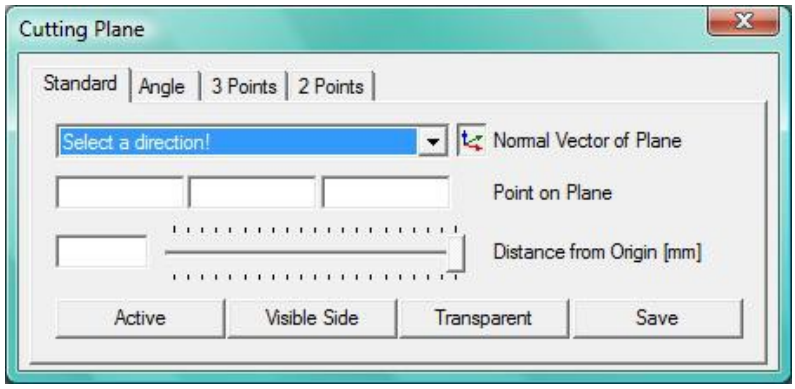
16.3.4 Cutaway

Often parts have areas not easily visible from outside. To access these areas, you can cut your parts with the function “Cutaway”.

(1) Click "Cutaway".

⇒ The window "Cutting Plane" appears on the screen.

Register - Standard



(2) Choose the normal vector of the cutting plane in the pull down menu or click on the coordinate axis, to enter the vector manually.

(3) Click the button "active" to activate the display of the cut.

(4) Move the slider to control the position of the cutting plane

or

define a point in the cutting plane by entering the coordinates or digitalizing a point on the surface of the part.

(5) Click "Visible Side" to change the visible side at the cutting plane.

(6) Activate "Surface" to show the cutting plane as non translucent surface.

On the following registers, you can define the cutting plane by other possibilities:

- Choose "**Angle**" to rotate the X-Y plane and move this plane into a certain point.
- Choose "**3 Points**" to define a plane by giving the coordinates of three points lying in the cutting plane
- Choose "**2 Points**" to define a cutting plane by giving the coordinates of the beginning and the end of the normal vector.

16.3.5 Font

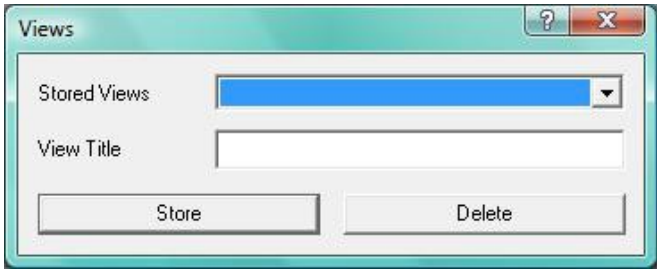
You can choose the font for the scale, the title and the name of geometry.

16.3.6 Save

You can store different views of your part. These are valid in the current directory.

Click "Save".

⇒ The window „Views" appears on the screen. Loading may take some time in a directory with many files.



16.3.6.1 Store Views

(1) Identify the view, which you would like to store, by entering the name into the field: "View Title".

(2) Click "Save".

⇒ The name of the selected view appears in the field "Stored Views", after the view is stored.

16.3.6.2 Select A View

(1) Click the arrow in the field "Stored Views", in order to select a stored view.

(2) Click on the selected view.

16.3.6.3 Delete A View

Mark a stored view and click "Delete".

16.3.7 Split Screen

You can compare results of different simulations of a part in split windows.
For this choose a screen splitting mode and open a simulation in every window. If you now open a result in the main window (not “Remote Controlled” in the status bar), this result will be displayed in all windows. Also, if you move your part in the main window, it will move the same way in all “Remote Controlled” windows.

16.3.8 Standard Views

With each of these functions you can choose from two standard views.

16.3.9 Zoom

Zoom allows to magnify specific areas of the display.

16.3.9.1 Full size

Click "Zoom -> Full Size" to project the full part on the screen.

16.3.9.2 Rectangle

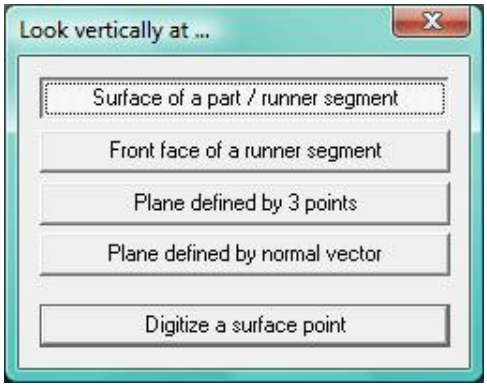
(1) Click "Zoom -> Rectangle".

(2) Drag open a rectangle on screen.

(3) Release the mouse button.

16.3.10 Look vertically at...

The exact definition of the aspect of your part is very important to position several parts in a mold.


Look vertically at ...
Surface of a part / runner segment
Front face of a runner segment
Plane defined by 3 points
Plane defined by normal vector
Digitize a surface point

1) Click < Look vertically at... >.

⇒ The window “Look vertically at...” appears.

You can define the line of sight in four different ways.

Look vertically at...

- Surface of a part / runner segment

  1) Push the button and select with “Ctrl + left mouse button” a surface of the part resp. of a runner segment
- Front face of a runner segment

  1) Push the button and select with “Ctrl + left mouse button” an end of a runner segment
- Plane defined by 3 points

  1) Push the button down and push afterward the button “Digitize 3 points”.

  2) You can now select three points on the surface of the part, which will define the plane.

  Please consider also the note to the “Point on... function”.
- Plane defined by normal vector

  1) Push the button down and push afterward the button “Enter the normal vector”.

  2) You can enter the normal vector in the window, that appears on the screen.

16.3.11 Mesh Statistic

The Quality of your CFE – Mesh can be checked.

- Area

Diagram: Area [mm²] (Triangular Elements [%])

Proportional value of the triangle elements of the surface size [mm²].

- Quality

Diagram: Quality [%] (Triangular Elements [%])

The evaluation of the element quality contains the criterion: **Smallest angle / 60°**.

The bigger the quotient, the better the element quality.

- Min. Angle

Diagram: Min. Angle [°] (Triangular Elements [%])

Proportional value of the triangle elements with the smallest angle [°].

Sum of the Angles = 180°

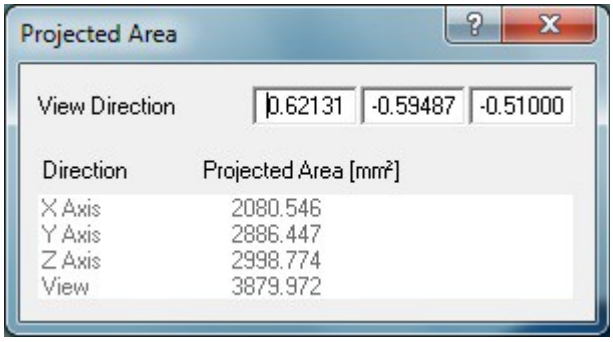
- Max. Angle

Diagram: Max. Angle [°] (Triangle Elements [%])

Proportional value of the triangle elements with the largest angle [°].

Sum of the Angles = 180°

16.3.12 Projected Area



These function shows you the normal vector of the screen plane and the projected area of your part in X-, Y- and Z- as well as in viewing direction.

16.3.13 Projection

With this functions you can switch between orthogonal and perspective view.

Switching between orthogonal/perspective and stereoscopic view while the program is running is not possible at the moment. Therefore you have to restart the program.

Please see also chapter: 3D-Display with stereo projection

16.3.14 Options

The display of your part on the screen can be altered.

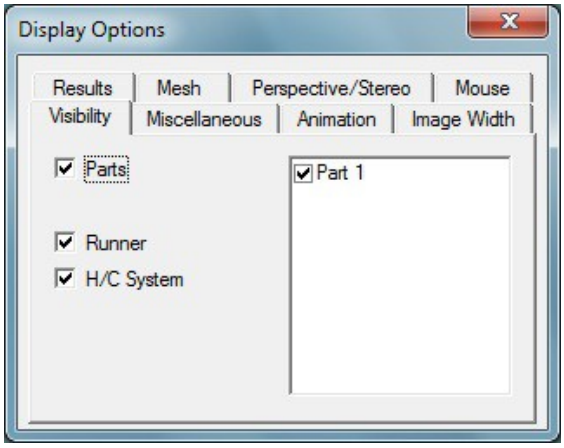
16.3.14.1 All

Click "Options->All".

⇒ The window "Display Options" with all tabs appears on the screen.

On the following commands you can call up every tab directly.

16.3.14.2 Visibility



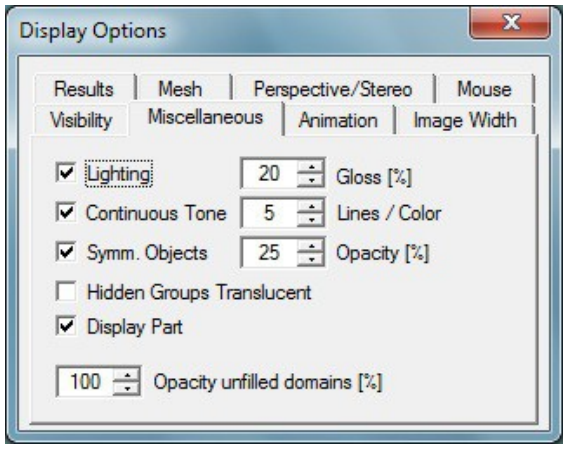
The checkmark (√) activates the “Visibility” of:

- Parts
- Inserts
- Runner
- Heating / Cooling System

Parts can be shown or hidden with the object list. A deselected object disappears immediately from other lists.

With the combination “Ctrl/Strg + left mouse button” you can digitalize objects. The digitalized object will be scrolled in the visible section of the object list and deselected. On this way you can make it invisible immediately.

16.3.14.3 Miscellaneous



16.3.14.3.1 Lighting

The checkmark (√) activates the „Lighting“. This effects that the part is displayed shaded.

16.3.14.3.2 Gloss [%]

You can affect the representation of the results on surfaces parallel to the screen by the function “Gloss [%]”.

16.3.14.3.3 Continuous Tone (continuous color)

The check mark (√) activates "Continuous Tone".
Otherwise, the colour gradient is only displayed as lines.

16.3.14.3.4 Lines / Color

You can define the way your results will be represented in "Lines / Color". This becomes active only if you have deactivated "Continuous Tone" before.

- Mark the displayed number and enter the desired number of lines per color by keyboard,

or

- click the arrows up or down, until the desired number of lines per color is displayed.

16.3.14.3.5 Symm. Objects

The checkmark (√) activates the „Symm. Objects”
(*See* chapter: Symmetry Check)

16.3.14.3.6 Opacity [%]

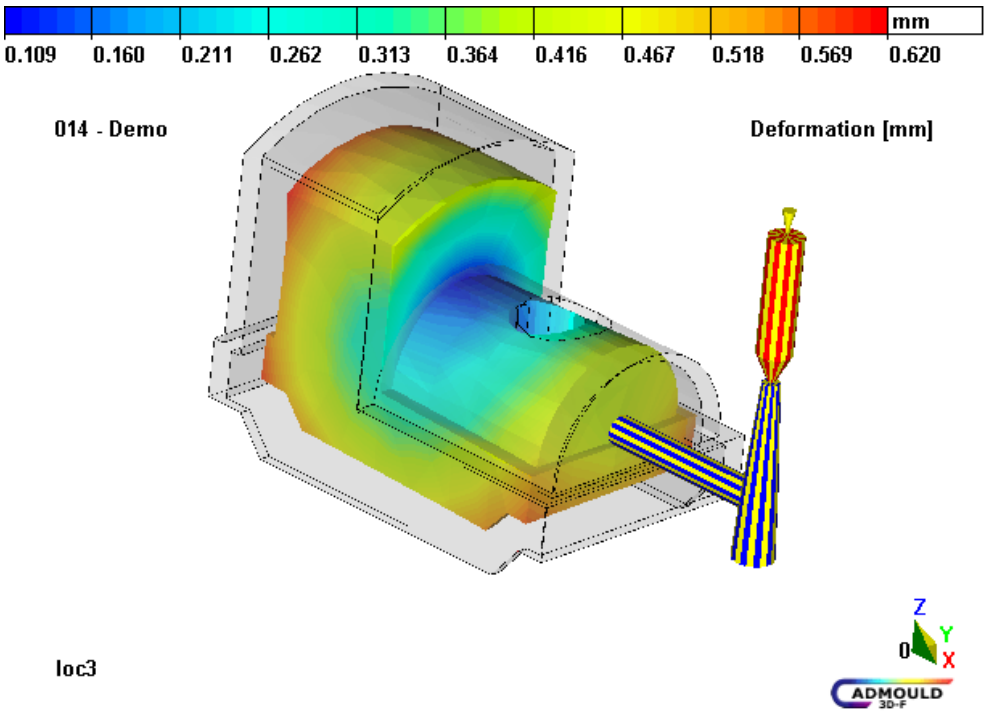
Here you can define the opacity of "Hidden Groups" or the original part in deformation results.

- Mark the number and enter a per cent value of the opacity

or

- click the arrows up and/or down, until the appropriate numerical value is indicated.

Example:



The warpage is exaggerated in this example.

The shade is opaque by 30% and shows the shape of the part without the deformation results.

(*See* chapter: Deformation (Shrinkage and Warpage))

16.3.14.3.7 Hidden Groups Translucent

If this Option is activated, hidden groups are not blanked completely, but shown translucent. This can be helpful to keep orientation in complex parts.

16.3.14.3.8 Display Part (geometry on / off)

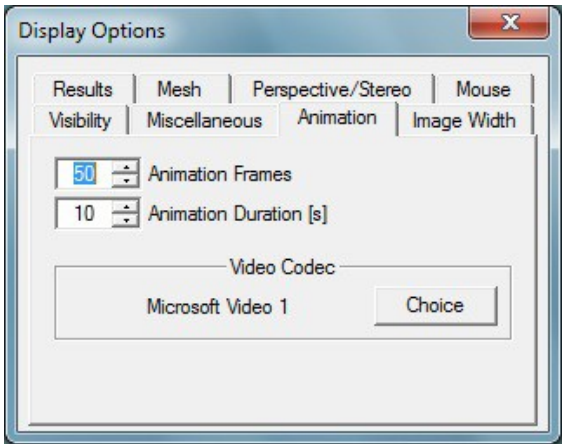
The check mark (√) activates the display of the part.

16.3.14.3.9 Transparency unfilled domains [%]

This function gives you the possibility to view the flow front in the inside of the part. For this choose a value smaller than 100%.

Please see also chapter: 3D-Flowfront

16.3.14.4 Animation



16.3.14.4.1 Animation Frames

Here you can pre-define the number of animation pictures.

- Mark the number and enter the number of pictures

or

- click the arrows up and/or down, until the appropriate numerical value is indicated.

16.3.14.4.2 Animation Duration

Here you can pre-define the duration of the animation.

- Mark the number and enter the time [s]

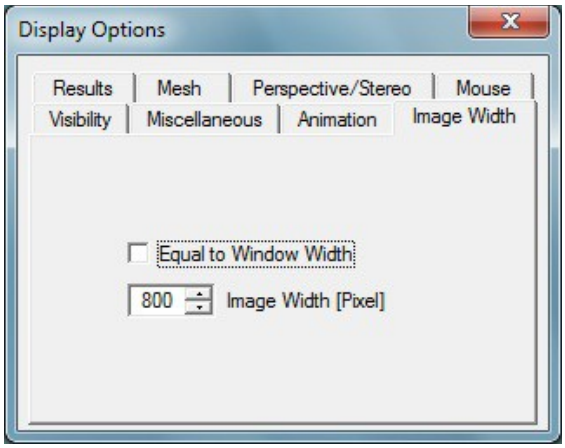
or

- click the arrows up and/or down, until the appropriate numerical value is indicated.

16.3.14.4.3 Video Codec

You can choose from different Video Codecs.

16.3.14.5 Image width in image files



Define the width of the pictures that are stored on your command or automatically while generating a report.

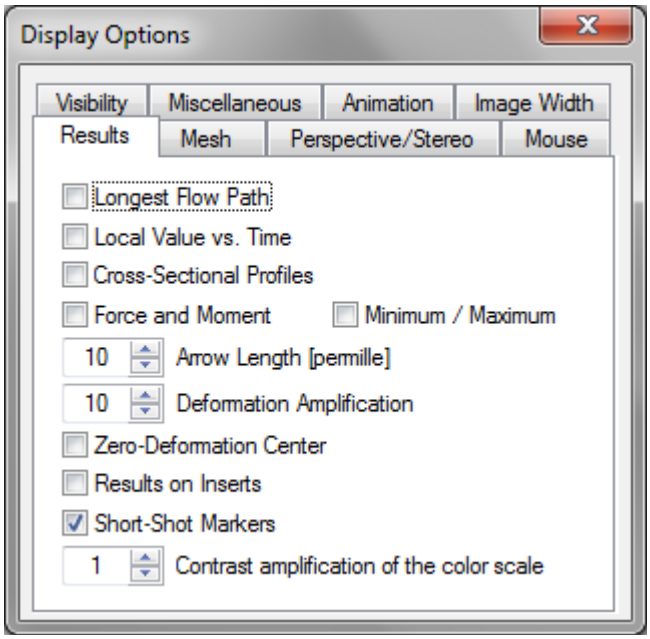
The checkmark (√) activates the „Equal to window width“.

- Click „Equal to window width“, in order to or deactivate it

or

- predefine the image width by entering the pixel number. The pictures will be recalculated in the background in the given size.

16.3.14.6 Results



16.3.14.6.1 Longest Flow Path

The check mark (√) activates the display of the longest flow path.

16.3.14.6.2 Cross-Sectional Profiles

The check mark (√) activates the display of diagrams over the profile, if saving of result in layers is activated.

You show a profile diagram by digitising results on the part (CTRL + left mouse button).

16.3.14.6.3 Local Value vs. Time

This function allows you to show Check points as postprocessor function (without setting it before). You can draw a diagram for a digitalized point on the surface of the part. Digitalizing local results on parts or runner systems of the files

101 - Snapshot Filling

111 - Snapshot Packing

122 - Snapshot Filling Temperature Profiles

123 - Snapshot Filling Speed Profiles

125 - Snapshot Filling Density in Layers

132 - Snapshot Packing Temperature Profiles

133 - Snapshot Packing Speed Profiles

135 - Snapshot Packing Density in Layers

- Snapshot Wall Temperature

a diagram pops up, which shows the course of the actual result detail versus time.

This setting will be saved in the registry.

This option does not exist in Cadmould® 3D-F Rapid.

16.3.14.6.4 Force and Moment

The check mark (√) activates the display of the results of "Force and Moment". If activated the resulting force arrow and the arrows of the moments are shown.

The diagrams can always be called up from the result selection.

16.3.14.6.5 Minimum / Maximum

The checkmark (√) activates the display of a blue (Minimum) and red (Maximum) pin on the part. This option is available in some result details, like temperature and pressure, in the wall thickness dialogue or similar.

16.3.14.6.6 Arrow Length (permille)

Here you can define the length of the velocity-arrows.

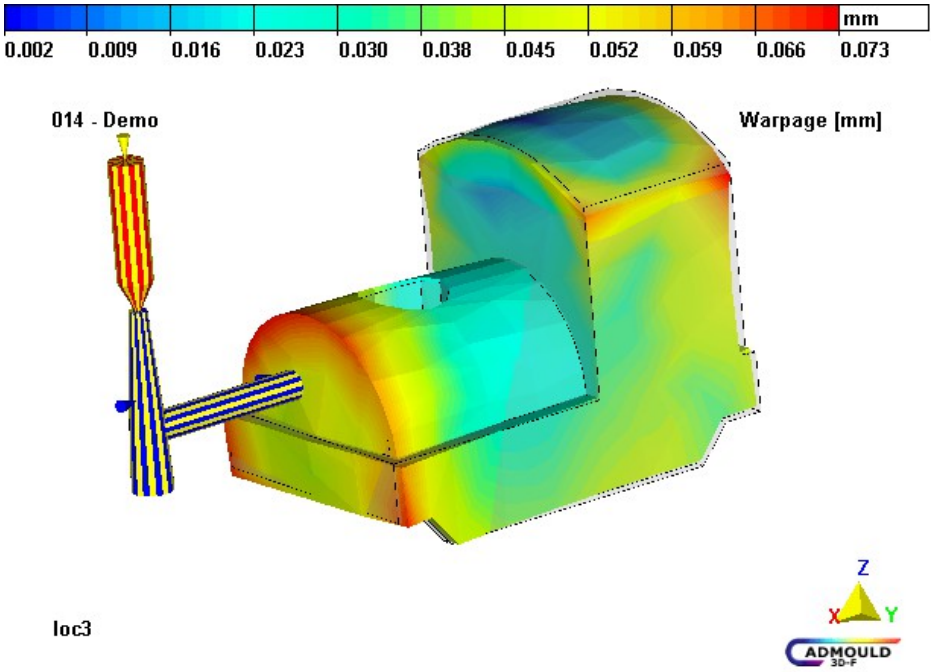
- Mark the number and enter length of the velocity-arrows
  or
- click the arrows up and/or down, until the appropriate numerical value is indicated.

16.3.14.6.7 Deformation Amplification

You can exaggerate the representation of the deformation results.

- Mark the number and enter the amplification of the deformation results representation
  or
- click the arrows up and/or down, until the appropriate numerical value is indicated.

Example:



The result "X-deformation" (the deformation in x-direction) is exaggerated.

16.3.14.6.8 Zero-Deformation Center

Activate this function to show a white ball at the zero-deformation center.

16.3.14.6.9 Results on Inserts

Activate this function to show also available results on inserts.

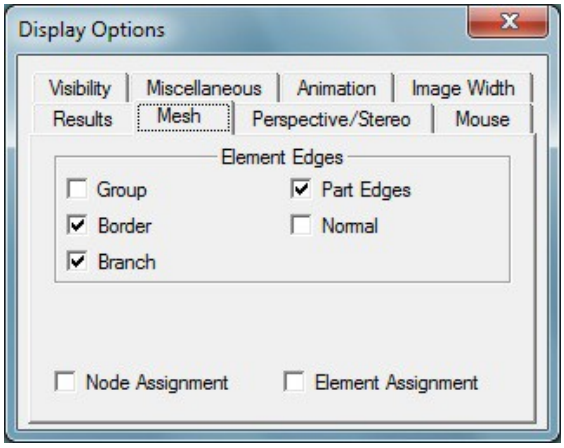
16.3.14.6.10 Short-Shot Markers

If the flow front is frozen during the filling simulation, the unfilled nodes are marked with blue balls. The unfilled nodes are displayed constantly when the checkmark is set here.

16.3.14.6.11 Contrast Amplification of the color scale

This integer (with the exception of zero) changes the normally linear relation between result value and color.

16.3.14.7 Mesh



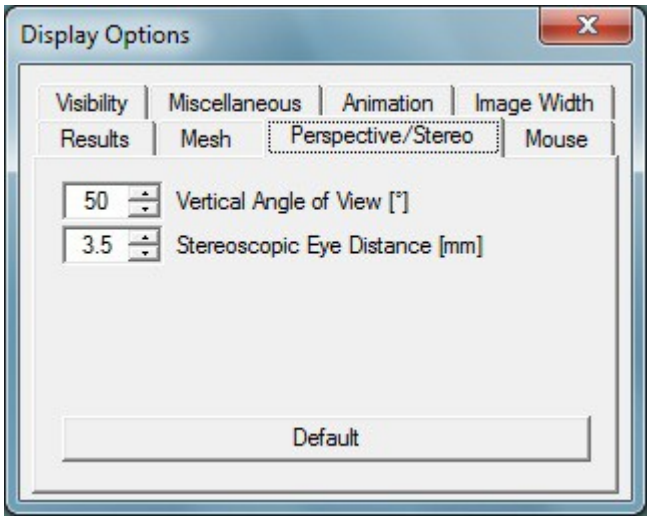
On this register you find all properties for the display of the FE-mesh.
Dependent on your license, you find here also the options node and element indices and assignment.

16.3.14.7.1 Element Edges

The checkmark (√) activates the display of the different “Element Edges“.

- Click "Part Edges" in order to activate or deactivate the part edges.
- Click "Normal" in order to activate or deactivate the display of the FE-mesh.
- Click "Group" in order to activate or deactivate edges, which split groups.
- Click "Border" to see element egdes on the border of the mesh. (Error)
- Click "Branch" to see element edges which connect more than 2 elements. (Error)

16.3.14.8 Perspective/Stereo



On this tab you can define settings for the perspective/stereoscopic view.

Vertical Angle of View [°]

This value defines the viewing angle for the perspective view.

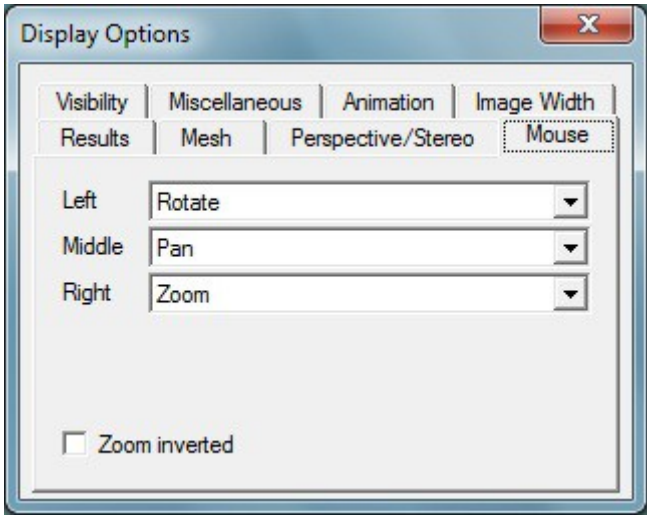
Stereoscopic Eye Distance [mm]

This value defines the eye distance for the stereoscopic view.

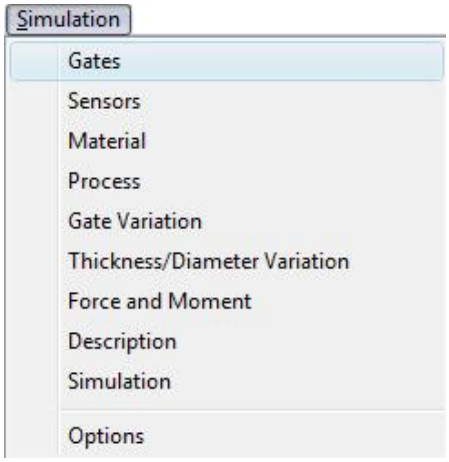
16.3.14.9 Mouse

On this tab you can define the function for each mouse button.

The function “Zoom inverted” inverts the direction of the mouse movement for zoom, as well as for the scroll wheel.



16.4 Simulation



Click "Simulation".

⇒ The pull down menu "Simulation" is activated.

In the pull down menu "Simulation" you find the following commands: Gate, Sensors, Shutoff Nozzles, Material, Process, Thickness/Diameter Variation, Force and Moment, Description, Simulation, CADMOULD® 3D-F Task Schedule and Options.

You can select these with a mouse-click or by keyboard.

(*See* chapter: "Mouse Control + Keyboard")

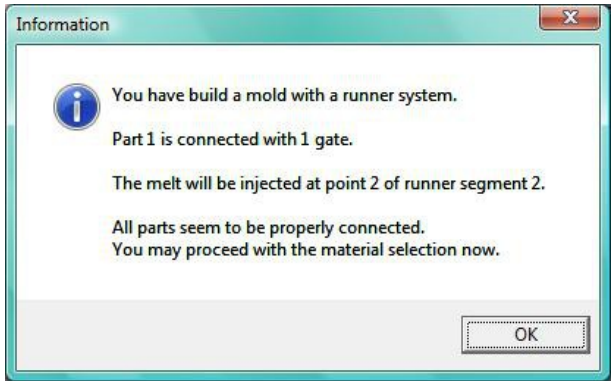
16.4.1 Gates

After the definition of a runner system you can check the connectivity of the mold and the runner system and the place of the injection point.

(1) Click "Gates".

⇒ The window "Information" appears on the screen.

(2) Click "OK", if all parts appear to be properly connected.


Information
You have build a mold with a runner system.
Part 1 is connected with 1 gate.
The melt will be injected at point 2 of runner segment 2.
All parts seem to be properly connected.
You may proceed with the material selection now.
OK

If you do not have defined a runner system or the injection point is not obvious, you have to define the coordinates of the gates.

(1) Click "Gates".

⇒ The window "Gate Locations" appears on the screen.

(2) Now, you can enter, edit, delete and optimize gate locations.
(For details please see Chapter: "Define a gate location")

(3) Click "OK" in order to accept the entered gates.

(4) Click "Cancel", in order to abort the input.

16.4.2 Sensors

The definition of sensors is analogical to the definition of gate positions.

See previous chapter and chapter: "Sensors".

16.4.3 Shutoff Nozzles

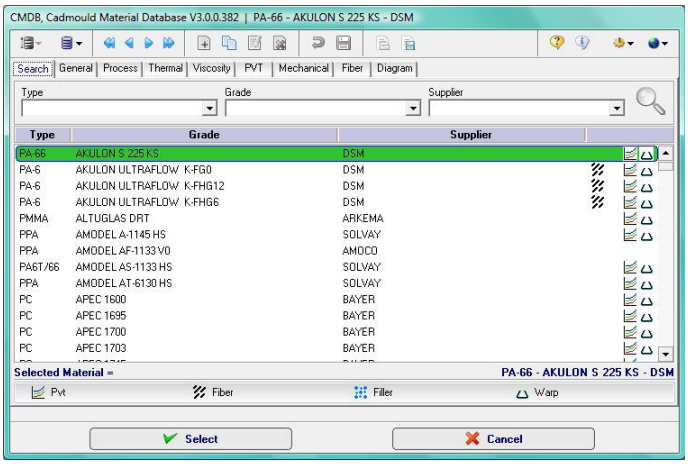
See chapter: “Cascade Injection Molding”.

16.4.4 Material

In order to define the material of the part, a delivery material data base is at your service. You can limit the search range, by entering the material name, the supplier or the material class.
You can create user-database with your own material. Therefore you can copy material data with the CADMOULD® 3D-F i0-format or in the Cadflow neutral-file format from the delivery data base or from user-database into your own user-database. The user-database can be extended with own material data.

16.4.4.1 Enter the material

(1) Click "Material".
(2) Wait, until the window "CADMOULD® 3D-F material data base" appears on the screen.
(3) Search for the material you want to use.
(4) Click on the material.
(5) Click "Select" to finish the selection.



Limit the search range

Seek after:
a) material class
(1) Click the arrow "Type".
- A list of the existing materials appears.

(2) Click on the material class of the material you are looking for.
⇒ All the materials of this material class are indicated.

b) material name
Type the initial letters of the material into the field “Material Name”.
- A list of the existing materials fitting this criterion is displayed.

c) the supplier

(1) Click the arrow "Supplier".
- A list of the existing suppliers appears.

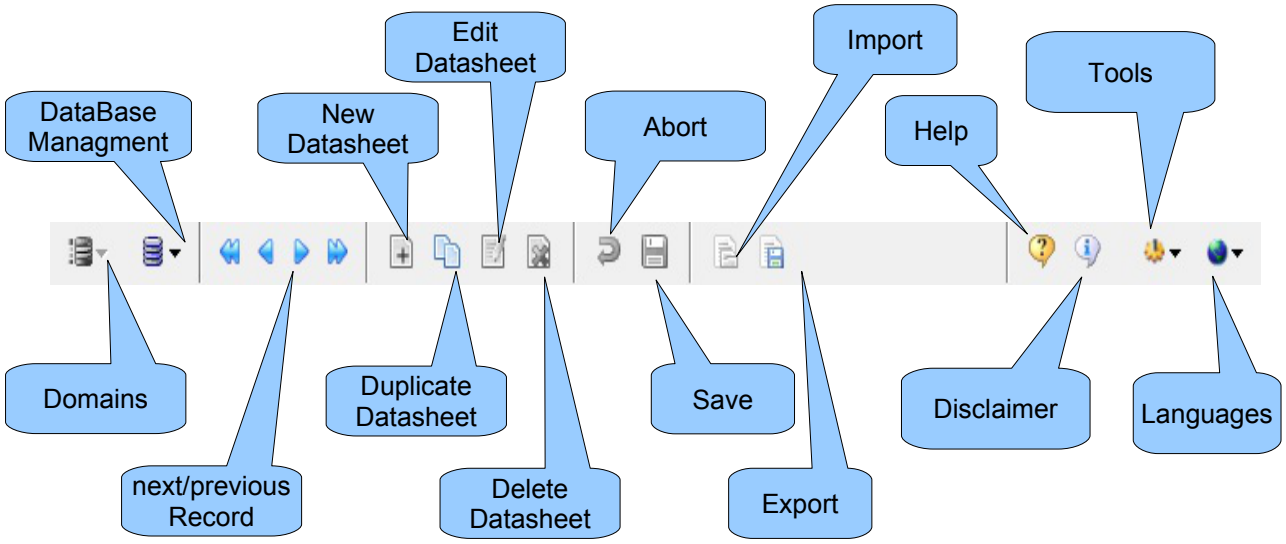
(2) Click on the supplier of the material you are looking for.
⇒ All materials of this supplier are indicated.

Caption:

The caption shows you the meaning of the symbols, which you can find in the last column of the table. These column shows, which data are available for this material.

16.4.4.2 Toolbox: CADMOULD® 3D-F material data base

16.4.4.3 Add your own material data into material data base.



If you want to add own material data, you have to create an user-database, because the materials in the delivery database can´t be edited. They can only be copied into the user-database and be edited there. Use therefore the function “Duplicate Datasheet”.

(1) Click "Material".

(2) Wait, until the window "CADMOULD® 3D-F Material data base" appears on the screen.

(3) Switch to the user-database, by button “Database management” or create a new one.

(4) Click "add new record".

⇒ The window "Material data" appears on the screen.

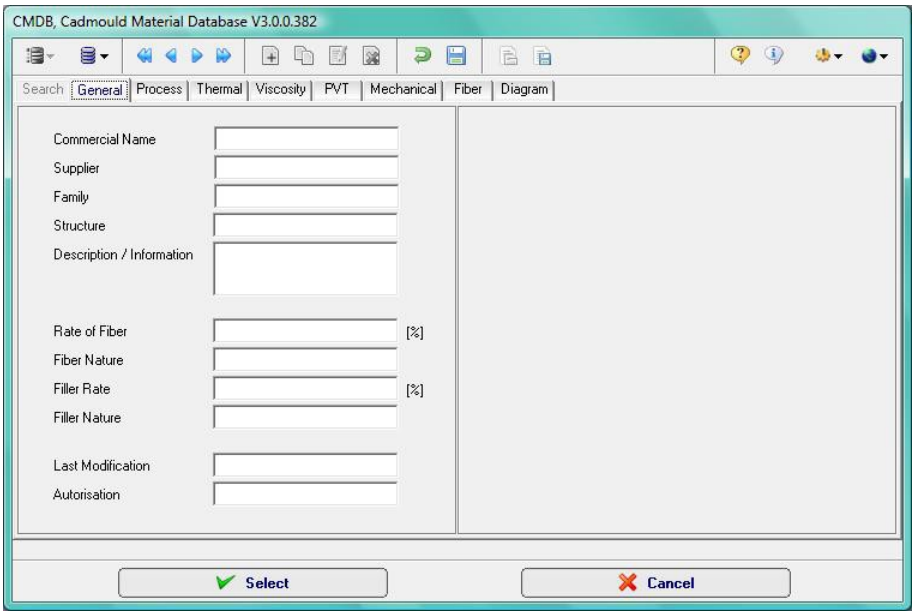
(5) Enter the material name, type, and supplier. Without these data the data record cannot be stored.

(6) Click on "Viscosity".

(7) Enter the Carreau- or Cross-Coefficients.

(8) Click "Thermal".

(9) Enter the thermal data.



Note: CADMOULD® 3D-F, CMSolver and RAPID can simulate with temperature dependent thermal material properties. The material data base is extended as well. The data will be saved in tables in the data base. If the chosen material has a temperature dependency, the program will use this data set, but only if pvT-data are present as well. The temperature dependent thermal material properties will be written in the dialog "Simulation Description".

(10)Click "PVT".

(11)Enter the pvT parameters using the IKV-, Renner- or 2-Domain-Tait-Ansatz.

(12)Click "Mechanical".

(13)Enter the mechanical data.

(14)Save the data record by clicking “save changes”

or

chancel your data by clicking “cancel changes”

During the input an examination of the input values takes place. Necessary inputs are:

- Name
- Type
- Supplier
- Rheological data
- Thermal data (conductivity, diffusivity and no-flow temperature)

16.4.4.4 Diagrams in the material data base

For the following material properties the appropriate diagrams can be represented in the data base:

- viscosity
- pvT parameters
- mechanical data

  (Young’s modulus and Poisson’s ratio)

Example:

Click on diagram and choose one, by selecting it from the list with the green arrow.

Note: Diagrams are only shown, if the optimal processing temperatures are entered on the register process.

16.4.4.5 Data exchange

The CMDB serves two possibilities to exchange data between different database or to dispatch material data.

16.4.4.5.1 Export

(1) Click "Material".

(2) The window "CADMOULD® 3D-F Material data base" appears on the screen.

(3) Select a material.

(4) Click on “Export”

(5) Give a file name and choose the file type: “Cadmould i0” or “Neutral File”.

(6) Save the material data.

The material data are saved as TXT-file (Cadmould i0) or as PLB-file (Neutral file, recommended, because in this file format all material data is saved) and can be imported into a user data base or be led further.

Note: Material data is a component of the CADMOULD® 3D-F software. Consider please the copyright regulations. Therefore the export function is only available, if the database is started out of the program window of CADMOULD® 3D-F.

16.4.4.5.2 Import

I0-files are TXT-files. Neutral-files are PLB-files. Both file types and can be imported into a CADMOULD® 3D-F user database.

(1) Click "Material".

(2) The window "CADMOULD® 3D-F Material data base" appears on the screen.

(3) Click on "add new record" or "edit this record".

(4) Click "Import".

(5) Open the appropriate file.

(6) The material date is read in into the input fields.

(7) Save the material data.

16.4.5 Process

With "Process" you define the process parameters.

(1) Click "Process".

⇒ The window "Process Parameters" appears on the screen.

(2) Define the Filling Time, Pressure Controlled Filling, Melt Temperature, Hot Runner Temperature, Wall Temperature and Ejection Temperature,

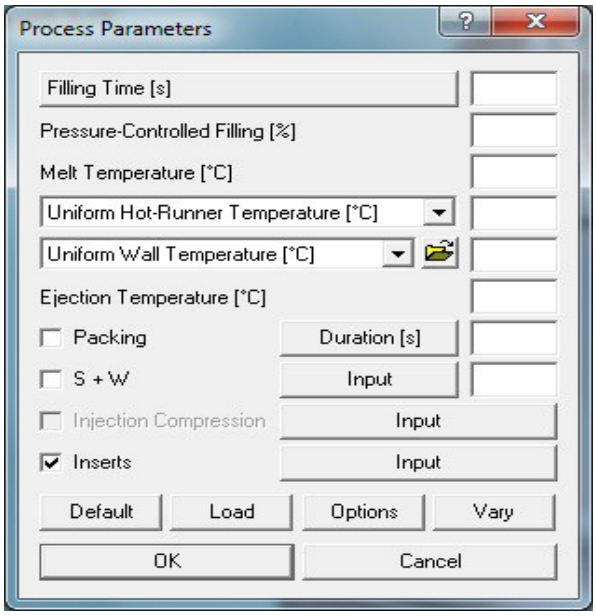
or

click "Default", if you want to use proposed values.

(3) Change the values, if you do not accept them.

(4) Click "OK", if you accept the parameters.

(*See* chapter: Define the process conditions)



16.4.6 Gate Variation

The function “Gate Variation” makes it possible for you, to compare different gating positions by variant calculations with each other.

(1) Click on "Gate Variation".

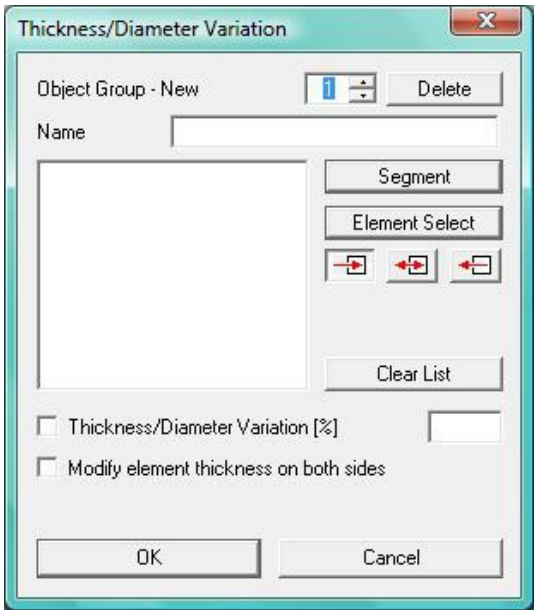
⇒ The window "Gate Variation" appears on the screen.

(2) You can now assign 2 variations for each gate. Therefore activate the checkmark in front of the gate you want to vary and select the positions “-1” and “1” by digitizing it on the part.

(3) You get now a line on the part, which shows the different positions.

If you now leave the window with “OK” and start the simulation, CADMOULD® 3D-F will automatically launch a variant calculation. (Module “Batch” must be licensed.)
Please see also chapter: Variation results

16.4.7 Thickness/Diameter Variation



This function allows you to vary the wall thickness an the diameter of runner- or heating/cooling-segments. It can be helpful to build flow aids, resp. to balance a multi connection cavity.

(1) Click on "Thickness/Diameter Variation".

⇒ The window "Thickness/Diameter Variation" appears on the screen.

(2) The definition of the elements you want to vary, is done with object groups.
(See also next chapter: "Object groups")

(3) Activate the option "Thickness/Diameter Variation [%]" for the object groups you want to vary and define the variation of the initial value.

⇒ If you activate the option "Change part shape as well", the CFE mesh will actually be varied one-sided e.g. double sided (option: "modify element thickness on both sides") by the defined variation. Thus the shape and the volume of the part will change.

(4) Click "OK" to finish the input.

If you now start the simulation, the needed variant calculations are written into the CADMOULD® 3D-F Task Schedule. (Only if module “Batch” is licensed. Otherwise you have to start the simulations from the new directory manually.) The number of the variant calculations depends on the number of object groups and is calculated with the "Taguchi" method.

Note: The combination of N object groups with 3 possible conditions, would cause 3^N simulations, if proceeded naive. When using a method, developed by Mr. Taguchi, the number of necessary simulations is reduced.

N	S	T
1	3	3
2	9	5
3	27	5
4	91	9
5	243	9
6	729	9
7	2187	9

You find the results of the variation in the menu <Results> under <Variation>.
(Please see chapter "Result Variation")

16.4.7.1 Object groups

Object groups are a combination of elements, runner- and heating/cooling segments. You can define as much independent object groups as you like. In the first line you can choose the object group you would like to define.

A new object group has always the first free index. Existing object groups can be edited or deleted with the button "Delete".
The indices of the following groups are than decreased by the value of groups you have deleted.
Depending on the dialog, there are different functions:

Segment selection:

The function "Segment" serves the selection of runner and h/c-segments. To choose segments click the segment with <Ctrl + left mouse button>. Selected segments are displayed red. You finish the choice with right mouse button.

Element selection:

The function "Element select" serves the choice of part elements. You can choose elements with the tools from the selection table. (See table: "Selection tools")
The chosen elements are displayed in the colour of the object group, as long as the dialog is opened.

Selection options:

This buttons define what happens to the selected elements or segments.

If the left button is pressed, all selected elements are added to the object group.

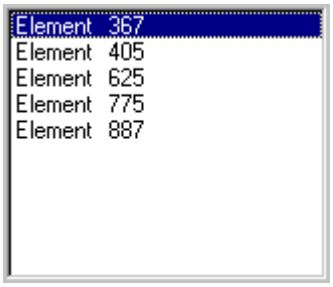
If the button in the middle is pressed, all elements which are already in the object group are deleted from it and all other are added to the object group.

If the right button is pressed, all selected elements are deletes from the object group.

Clear List:

Press the button "Clear List" to delete all elements and segments from the group, but the object group still exists.

Element List:



In the list you see all elements and segments that belong to the group. You can delete several or all elements with keyboard or context menu (right mouse click). When you select an element in the list, it is marked red on the part.

16.4.8 Force and Moment

(not included in CADMOULD® 3D-F RAPID)

This function allows it to calculate forces and moments on defined surfaces. You can for example calculate the forces on a slider or core.

As result you get the resulting force, as well as diagrams which displays the development of the forces and moments over the time.

Click in the menu "Simulation" on "Force and Moment".

⇒ The window "Force and Moment" appears on the screen.

(1) Define an object group, which contains the elements of the surface, for which you want to calculate the forces and moments.

(2) Define the coordinate system for the forces and moments. The coordinate origin is also the point of load incidence.

For the definition of the coordinate system, you have to define the origin, as well as one point on the X axis and one point on the X-Y plane.

(3) Activate the option "Determine force and moment" for the object group.

(Please see also chapters: "Object groups" and "Example: Force and Moment")

16.4.9 Description

In "Result Identifier" you must enter the identification number, under which the simulation data are stored or use the system default. You must describe the simulation with some key words in “Title”. You can give further explanations about your simulation in “Description”.

(1) Click "Description".

⇒ The window "Simulation Description" appears on the screen.

(2) Examine the path in which the results are stored.

a) They are OK

⇒ Continue with point (3).

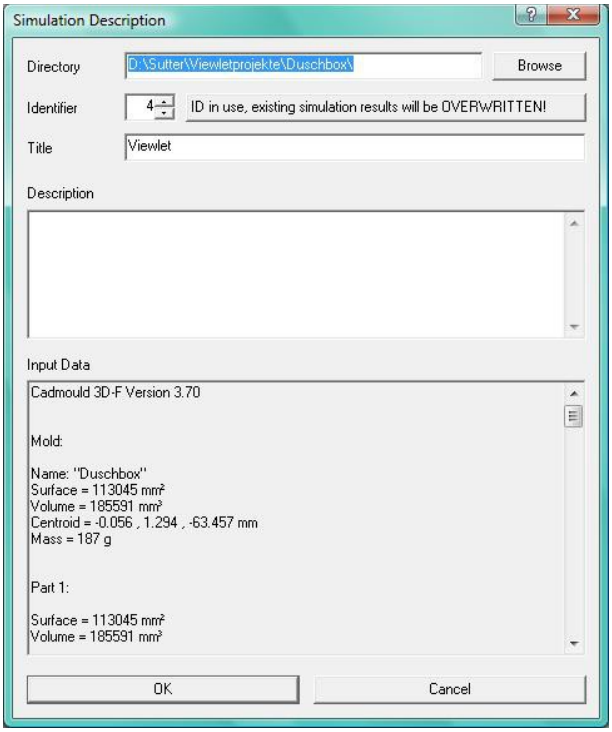
b) They are not OK

1) Click "Browse".

⇒ The window "Browse for Folder" appears on the screen.

2) Define the proper directory.

3) Click "OK".



(3) Enter the identification number in "Result Identifier".

The button to the right of the input field indicates the state of the selected number. Clicking on it selects the lowest unused number.

(4) Enter a short name for the simulation run in "Title".

(5) (optional) Describe the simulation process in the field "Description".

(6) Click "OK".

16.4.10 Simulation

You start the simulation run of the injection molding by clicking "Simulation".

Before you start the simulation, please consider the chapter: "**Options**"

(1) Click "Simulation".

(2) Wait until you get a message that the simulation is completed.

While the simulation is running intermediate results will appear on the screen.

16.4.11 CADMOULD® 3D-F Task Schedule

See chapter: “Stack and Compute Projects”.

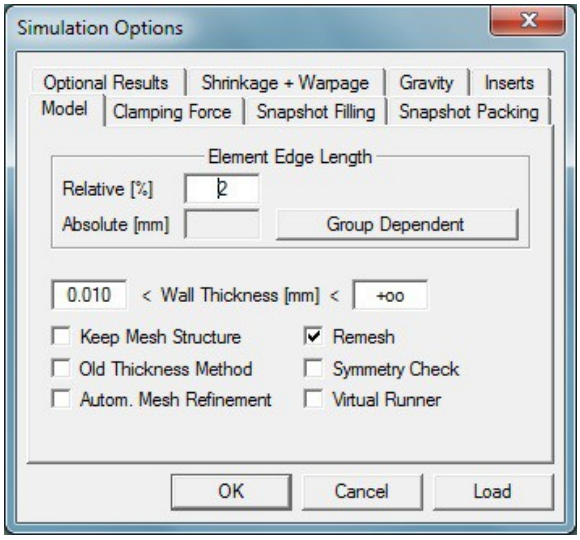
16.4.12 Options

(1) Click “Options”

⇒ The window: “Simulation Options” appears on the screen.

Note: When you press the button "Load", you can optionally load the settings for the current page or for all pages from an existing project.

16.4.12.1 Model



16.4.12.1.1 Element Edge Length

Enter a „Relative Length [%]“ (between 0.1 and 10) or an “Absolute Length [mm]” to define the accuracy of the results. The smaller the value, the more exact the results, however, the number of elements and the computing time increases.
The relative length is defined by the body diagonal of the bounding box of your part.

16.4.12.1.2 Wall Thickness

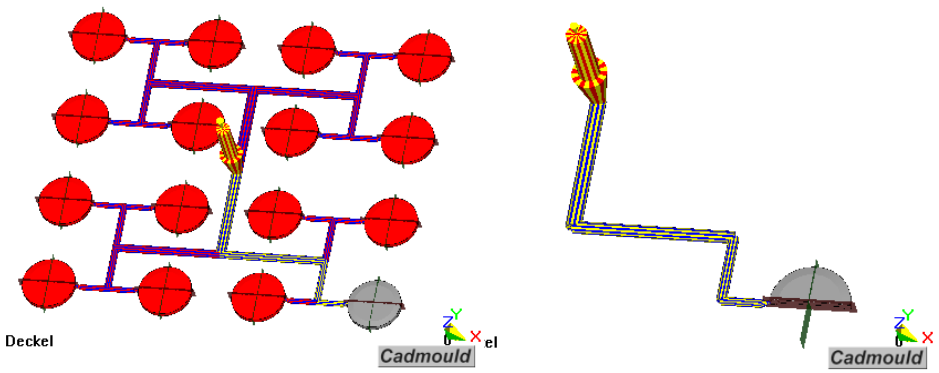
Enter a minimum and maximum wall thickness of the part. The limitation of the wall thickness is to prevent that wall thickness is not generated long the part.

16.4.12.1.3 Automatic Mesh Refinement

The activation of "Automatic Mesh Refinement" leads to more exact results, however, the number of elements and the computing time increases.

16.4.12.1.4 Symmetry Check

This is an alternative method to improve symmetries. It is different to the manual defined method, because the whole model must be build first. This can cause memory trouble.
Therefore the manual proceeding is recommended.
Both procedures reduce the calculating time extensive.
CADMOULD® 3D-F recognizes automatically symmetries of your part and of the runner system.
The congruence of the multiple cavities is recognized and used.



You are using the symmetries for the simulation by activating the Symmetry Check and confirming it after the gate definition.
The symmetries are red marked. (*See* chapter: Symm. Objects)

16.4.12.1.5 Old Thickness Method

The algorithm for wall thickness determination was changed some versions back. Calculating on the old wall thickness method may be recommended infew cases.

16.4.12.1.6 Keep mesh structure

It's possible to keep the structure of an existing triangular mesh after the import in CADMOULD® 3D-F. CADMOULD® 3D-F will only prepare the wall thickness and the inner structure. All triangular elements will be kept.

16.4.12.1.7 Remesh

During remesh, connected areas of elements lying in the same plane are identified. The boarding element edges of such an area remain, whereas the inside will be remeshed with elements of better quality.

Note: The option remesh can cause longer calculation times.

16.4.12.1.8 Virtual Runners

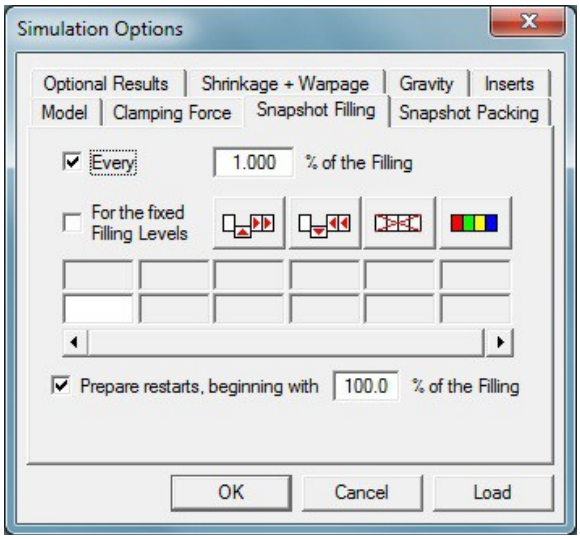
This option changes the distribution of the defined flow rate to the different defined runner positions.
If this option is not activated, the flow rate will be evenly distributed to the runner positions. Thus it can come to different pressures at the runner positions during the fill process, e.g if the mould has different wall thicknesses.
Especially in the last phase of the mold filling it can give a high pressure rise, caused by imbalance, if the material is pressed into an already filled section. In reality, the material will evade over another section.
By activating this function you can make allowance for this behavior. From now on one pressure level will be held at all runner positions. That will cause, that each runner position has another flow rate, which complies in summation to the defined value.
To attain a realistic result, the function "Virtual Runners" is activated by default.

Note: If you define 2 or more gate positions in the window "Gate-Positions", this function will automatically be activated and the gates appears as green cones on the display. You can deactivated this function in this dialogue here, if you want.

16.4.12.2 Clamping Force

Using symmetries, you can take into account undercuts for the calculation of the clamping force. This can extend the simulation time significantly. In this case it is advised to calculate without undercuts and take into account the real situation by multiplication.

16.4.12.3 Snapshot Filling



A result storage interval and fixed filling levels can be defined.

- Result storage interval

Every ...% of filling

- fixed filling levels

Entering of fixed values

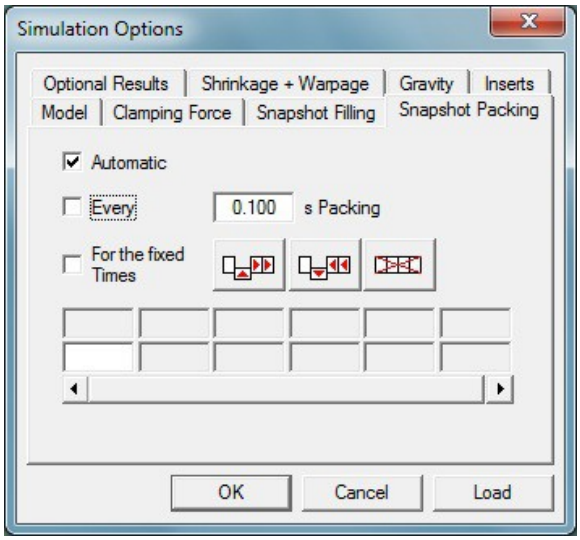
the Buttons from left:

insert column, delete column, all columns delete, insert default values.

- **Prepare restarts, beginning with X % of the filling**

  If this option is set, restart files will be written from this stage on together with the snapshot results. They have the extension ".rst". Hitting the "Go" button they will be offered in case of fitting to the running project. If you don't want to use a restart file, just click the "Cancel" button. The simulation will start normal from the beginning on.

16.4.12.4 Snapshot Packing



A result storage interval and fixed filling levels can be defined.

- Automatic
- Result storage interval

Every ...s packing

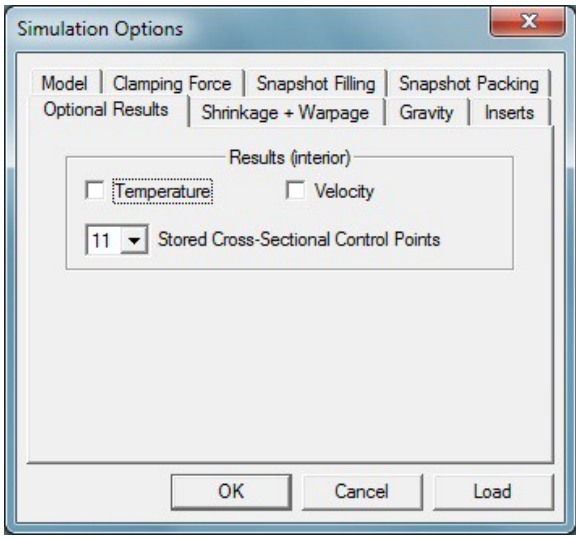
- fixed filling times

Entering of fixed values

the Buttons from left:

insert column, delete column, all columns delete, insert default values.

16.4.12.5 Optional Results

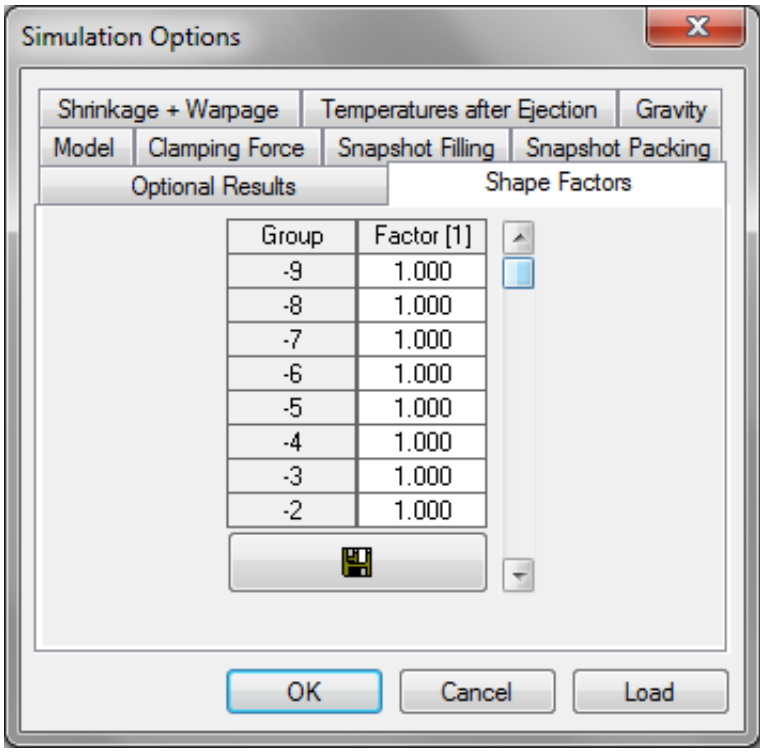


The interior temperature and velocity can be stored. The temperature distribution and a velocity profile can be seen for different layers over the wall thickness or been displayed on cutting planes.

Additionally define the number of control points which are stored over the profile.

(*See* chapter: "Result details")

16.4.12.6 Shape Factors



Motivation and definition of the term

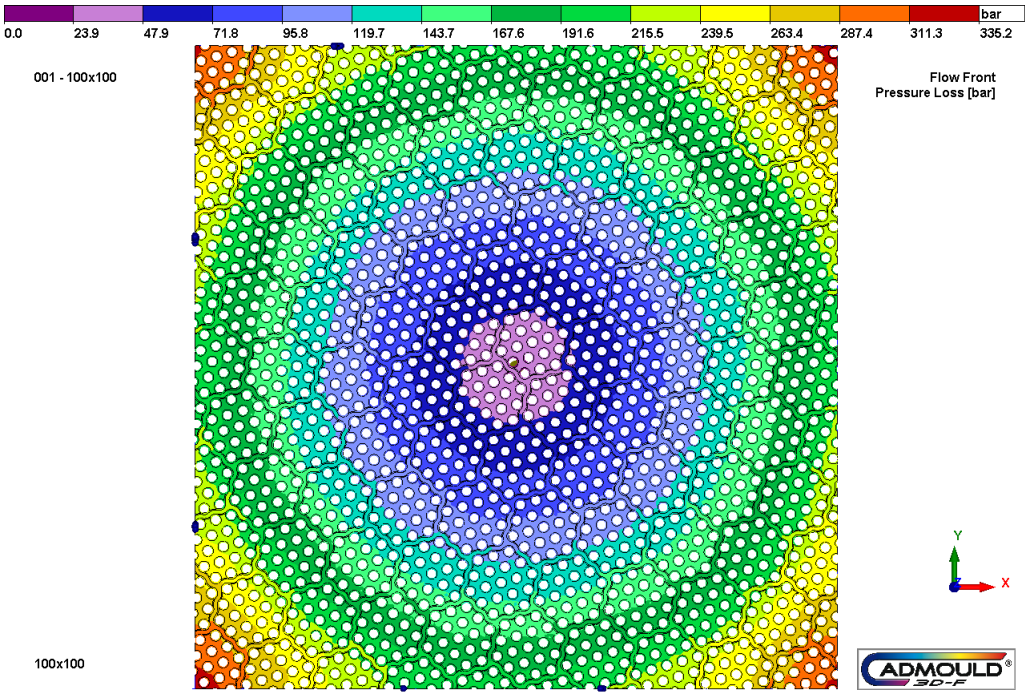
If complex shapes have to be simulated the number of elements and the simulation time may grow beyond tolerable limits. Then some sort of geometric modeling is required. In the case of tubular runner channels one would like to model a complex cross-sectional shape by a simple circle, described only be its diameter. If it comes to tiled or banded structures of part surfaces a simple plate of constant thickness would be optimal. A good start is to model for equal volume first. The pressure loss when filling, however, would required a different diameter or thickness respectively. The thickness attribute of the underlying rod or triangle elements is not sufficient in order to model both volumetrically and hydraulically correct. To resolve this dilemma a further element attribute is added, the shape factor. It is defined as the ratio of hydraulic thickness to volumetric thickness.

Shape factors for the modelling of 2-dimensional patterns

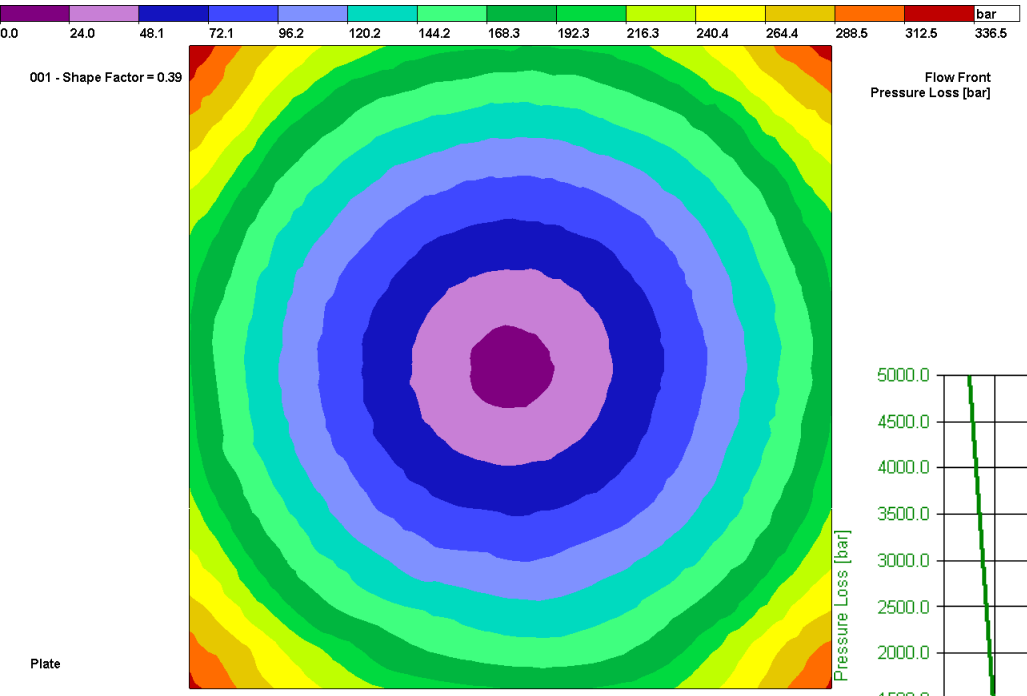
are best determined by experiment, in CADMOULD® 3D-F of course.

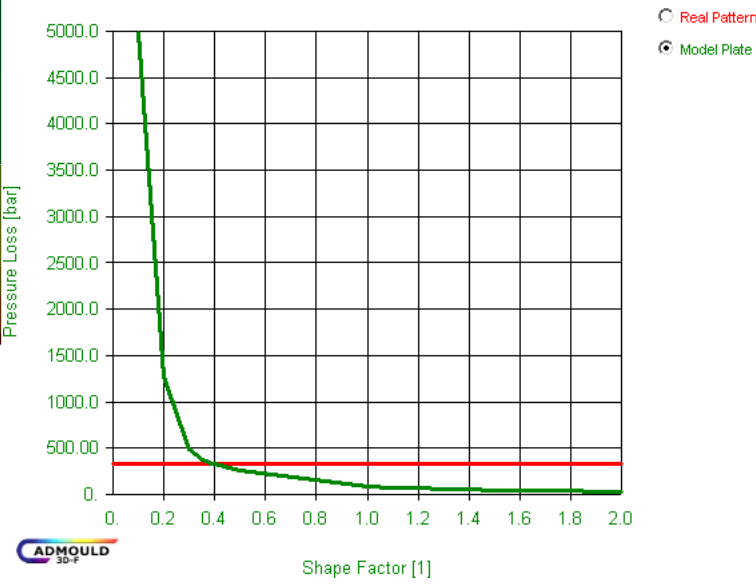
Here is a suggested procedure:

1. Cut out a square piece with a sufficiently hig number repetitions of pattern tiles and store it as a water-tight STL, with borders closed
2. Import the STL file into CADMOULD® 3D-F, set a single injection point in the middle, select the plastic material of your project, enter an adequate filling time and a sufficiently small element size. Then specify the simulation identifier and lunch the filling simulation (no packing required)

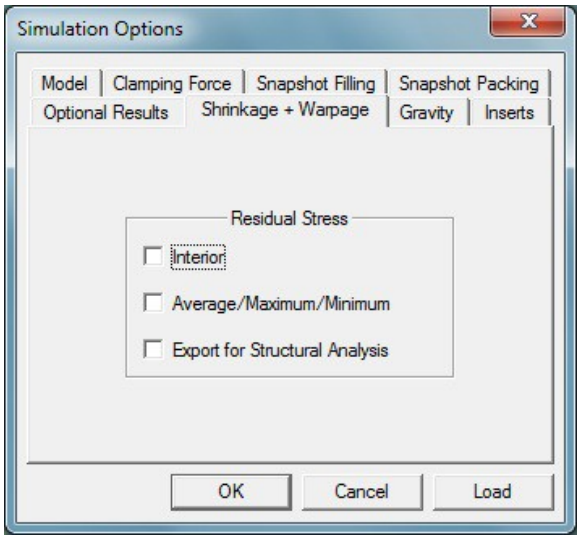


3. As soon as the mesh preparation is ready and the result-selection dialog appears, open the dialog “Simulation Description” and look for the volume of the part.
4. Create a STL file with a square flat plate of the same volume as the patterned part, i.e. with equal edge length and the equivalent thickness = $Volume_{patterned\ part}/EdgeLength^2$. Alternatively create a square midplane mesh with the equivalent parametrical thickness.
5. If the simulation of the patterned part is through display the result “Flow Front – Pressure Loss [bar]” and write down the maximum pressure from the right edge of the color scale.
6. Load the model part, enter a shape factor from the range 0.1 – 1.5 for element group 1, run an equivalent filling simulation and inspect the result “Flow Front – Pressure Loss [bar]”. Vary the shape factor until the pressure loss matches the one calculated for the patterned part.





16.4.12.7 Shrinkage and Warpage (Residual Stress)



- Interior
- Average/Maximum/Minimum
- Export for Structural Analysis

This file contains for all elements and all layers the three independent values of the stress tensor, related to the local element co-ordinate system.

The name of the file is /geometry name/ identifier of the filling simulation/.str (e.g. geom_001.str).

The file is binary and all values are single precision.

The order of storage is the following:

Number of nodes, number of elements of the geometry.

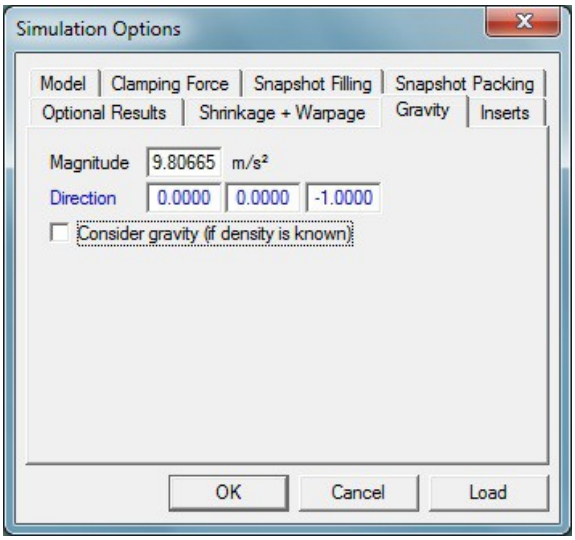
Diagonal value of x, diagonal value for y, x-y value.

First all values for all layers of one element are stored and then the values of the next element.

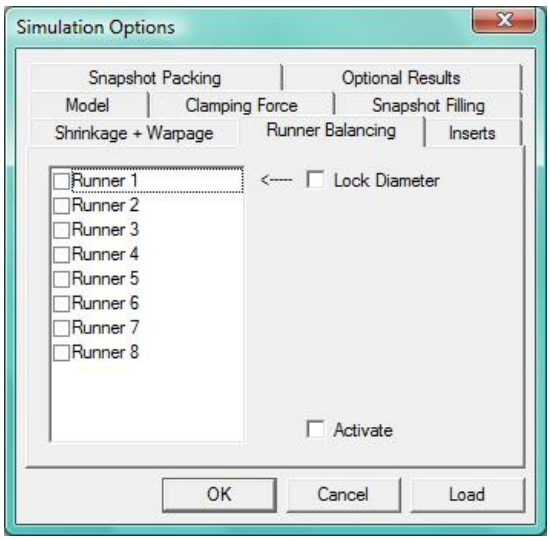
If the option "In Layers" is activated, these results can also be displayed as diagram over the profile.

16.4.12.8 Gravity

You can activate the consideration of gravity for the simulation. For this the density of the material must be known. The fields “Magnitude” and “Direction” have to be filled in. It may be necessary that you adjust the direction to your project.



16.4.12.9 Runner Balancing

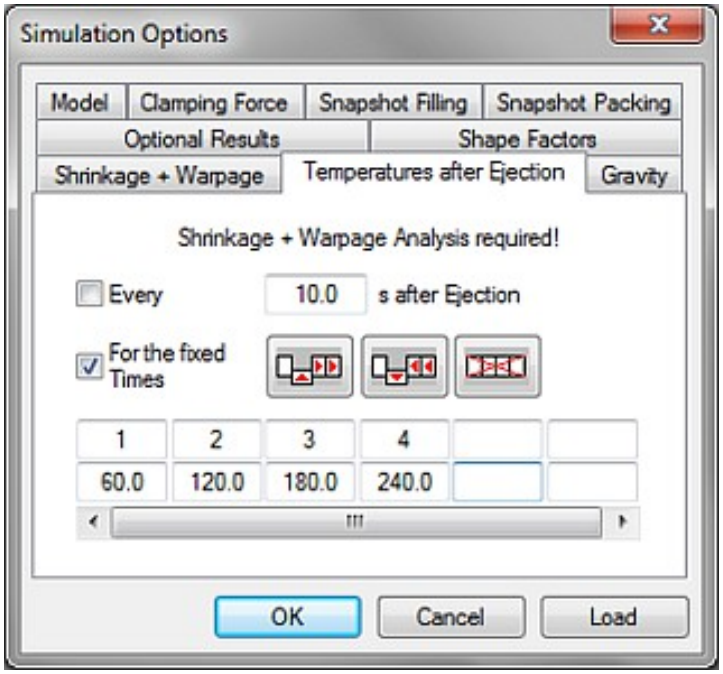


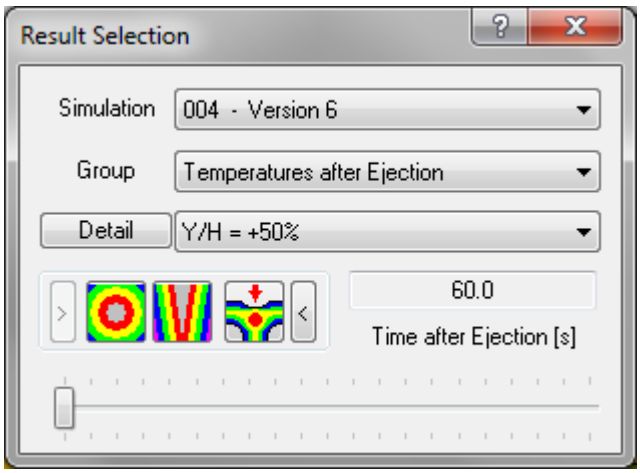
CADMOULD® 3D-F does an automatic runner balancing for family moulds if the option is activated. The program analyses the runner system and finds out, which diameter can be changed. During the balancing the material properties and the process will be considered. We recommend to start with bigger runner diameters, because they will be mainly reduced. You can define runner segments, which are not allowed to be changed.

Note: If you want to balance moulds with multi connected parts, which are already balanced, it is necessary to lock all diameters, that could influence the balance of the several parts.

16.4.12.10 Temperatures after Ejection

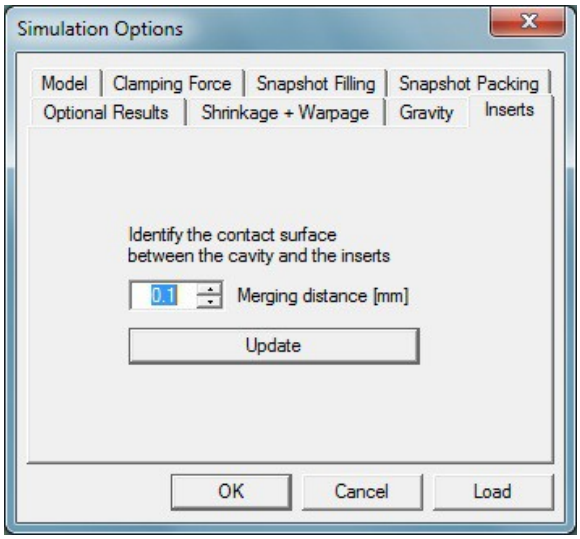
Between ejection time and final cooling optionally temperature snapshots of mould and inserts can be calculated. To have these extra results, choose a storage mode on the page "Temperatures after Ejection" of the dialog "Simulation Options". The temperatures are calculated during the shrinkage+warpage analysis, for each element at 11 sampling points across the cross-section and stored in files with the result identifier 118. The 7-digit time in the filename is in seconds, in contrast to other results, where it is milliseconds.





The temperature results will be offered as result group "Temperatures after Ejection", with the detail ranging from "Y / H = +50" to "Y / H = -50". For a comparison with images of a thermal camera please select "Y / H = +50" (surface temperature). Node-based display is recommended. These temperatures can be displayed and stored as an animation if more than one snapshot is specified.

16.4.12.11 Inserts



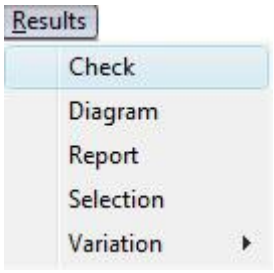
The register “Inserts” serves the control and the determination of the contact surface for 2K or insert simulations.

You can define the group, in which elements on the contact surface are sorted and the merging distance to identify contact surfaces.

Click “Update” to start the identification process. The older group allocation of the contact surface will be overwritten.

Note: Please consider also the chapter Inserts, as well as the separate manual 2K&Insert.

16.5 Results



Click "Results".

⇒ The pull down menu "Results" is activated.

In the pull down menu "Results" you find the following commands: Animation, Check, Diagram, Variation, Report and Selection

You can select these with a mouse-click or by keyboard.

(*See* chapter: "Mouse Control + Keyboard")

16.5.1 Animation

These functions allow to start, stop and pause an animation.

16.5.2 Check

The completeness of the results of a Project can be examined automatically.

16.5.3 Diagram

You can load and edit diagrams, without have loaded a project.

16.5.4 Report

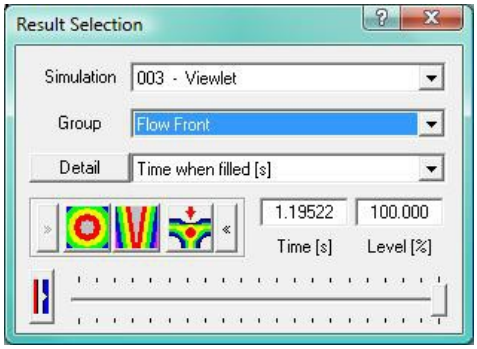
See chapter: "Report Filling Analysis"

16.5.5 Selection

With the instruction "Selection" you have the possibility to select the results for evaluation as soon as the simulation is finished.

(1) Click "Selection".

⇒ The window "Result Selection" appears on the screen.



(2) Click the arrow "Simulation".

(3) Select a simulation result by clicking it.

(4) Click the arrow "Result Group".

(5) Select a result category by clicking it.

(6) Click the arrow "Result Detail".

(7) Select the desired result by clicking it.

View the stored snapshot results. Press the "interpolation" button in the bottom left corner of the result selection window, and the slider will jump only to stored snapshot results (eg. each 10%).

Remark: Only snapshot results!

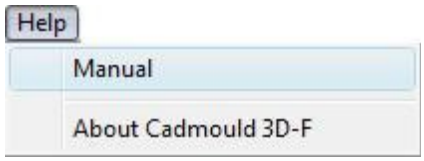
(For more details see chapter: "Result selection" and "Description of results")

16.5.6 Variation

This function contains the results of the process parameter variation.

For a description of the individual results please see chapter "Variation Results".

16.6 Help



Click “Help”.

⇒ The pull down menu ”Help“ is activated.

In the pull down menu “Help” you find the following instructions: “About CADMOULD® 3D-F” and “Manual“. You can select them by mouse-click or by keyboard.

(*See* chapter: “Mouse Control + Keyboard”)

16.6.1 Manual

(1) Click “Manual” to look up the help.

⇒ The manual appears on the screen as a PDF-file.

(2) Search your help topic from the table of contents (manual: *page:* 2) or from the reference register.

16.6.2 TeamViewer

TeamViewer is getting installed automatically with CADMOULD® 3D-F on your PC. With this software you can show your screen contents the CADMOULD® 3D-F support team. For it, you get an ID and a password from TeamViewer which you submit to the support team by phone.

16.6.3 Cadmould Wiki

Click on “Cadmould Wiki” to enter the site http://cadflow.fr/wiki .

16.6.4 About CADMOULD® 3D-F

Click “About CADMOULD® 3D-F” to get some information about CADMOULD® 3D-F, e.g. the version number and addresses.

17. Controls

17.1 Button Bar

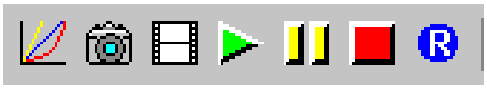
The buttons represent the most important commands of CADMOULD® 3D-F. They are organized in groups. Click the individual buttons, in order to select the command you want to use.

These buttons contains the commands: "Open", "Print", „About CADMOULD® 3D-F“, „Manual“. These corresponds to the same instructions: "Open", "Print" in the pull down menu: "File" and „About CADMOULD® 3D-F“, „Manual“ in the pull down menu: "Help"

(*See* chapter: Open, Print, About CADMOULD® 3D-F, Manual)

These buttons contain the commands: "Import STL", "Runner", "Gates", "Check Points", "Shutoff Nozzles", "Material", "Process", "Description", "Simulation", "CADMOULD® 3D-F Task Schedule". These correspond to the same commands: "Import STL", "Gates", "Material", "Process", "Description" and "Simulation **(Go)**" in the pull down menu: "File", “Edit” and "Simulation".

(*See* chapter: Import, Gates, Material, Process, Description, Simulation)



These buttons contain the commands: "View Results", “Save Window”, “Animation File”, "Play", "Pause", "Stop" and "Report". "View Results" and “Report” correspond to the commands: "Selection" and “Report” in the pull down menu: "Results", “Save Window” in the pull down menu: “File”.

(*See* chapters: Selection, Save Window, Animations Report)

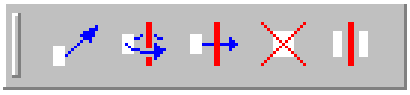
Toolbar: **Wall Temperature**

Pull down menu: *View/Toolbars/***Wall Temperature**

These buttons contain the commands: "H/C System from File", "Heating/Cooling System", "Heating/Cooling Circuits",

“Save Heating/Cooling System", “Options Wall Temperature Analysis”, “Wall Temperature Analysis”.

(*See* chapter: Toolbar "Wall Temperature")

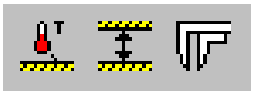


Toolbar: **Edit**

Pull down menu: *View/Toolbars/* **Edit**

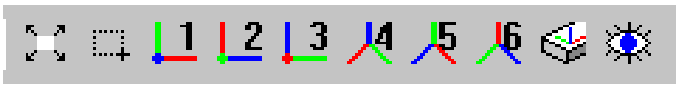
These buttons contain the commands: "Copy", "Rotate", "Mirror", "Delete"an "Cut".

(*See* chapter: Toolbar "Edit")



This button contains the command: “Wall Temperature”, “Wall Thickness” and “Measuring Devices”. This corresponds to the same commands: “Thickness” and “Measuring Devices” in the pull down menu: “Edit“.

(*See* chapter: Wall Temperature distribution, Wall Thickness, Measuring Devices)



The first button projects your part full size onto the screen. The second button launches the command: "Zoom".

(*see* chapter: Zoom)

The numbered buttons 1 to 6 contain 12 pre-defined views of the part. Each button contains two views of the part. The views can be chosen by clicking a button ones or twice. Different colors are assigned to each axis (x, y, z). The x-axis is red, the y-axis is green and the z-axis is blue. These functions give you a clear definition of the spatial position of the part.

The last two buttons contain the instructions: “Look vertically at...“ and "Save" in the menu "View".

(*see* chapter: Look vertically at..., Save)

17.2 Mouse Control + Keyboard

17.2.1 Combination of keys

You can select many instructions of CADMOULD® 3D-F by a combination of keys. Keep the "Alt" button pressed and press the underlined letters to call the command. If a combination of keys (“hot-keys”) is noted in the pull down menu behind the command, you can select the instruction directly without having to invoke the menu first.

Example:
"Zoom"
You find the command "Zoom" in the pull down menu: "View". Combination of keys: "Alt" + "v" + "z ",

Or

Combination of keys: "Ctrl" + "a"

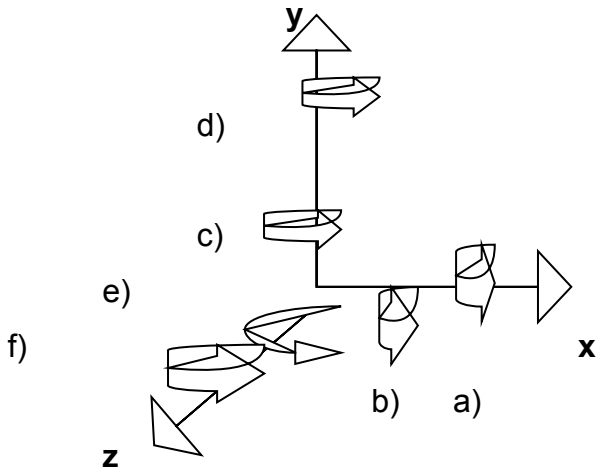
17.2.2 Light

New position of the light
Keep the keys <Ctrl> and <Shift> pressed. Click the left mouse button, keep it pressed and define a new position of the light by mouse.
Press the keys <Ctrl> + <Shift>, in order to activate the standard lighting.

17.2.3 Rotation

F3-key:

(1) Switch to the command <Rotation> in the status bar by pressing the F3-key.
(2) The part can be turned with the arrow keys and <PgUp> and <PgDn> around the axes parallel to the screen edges or with the < x, y, z > keys around the axes of the coordinate system now.
   a) **x-key**: the part turns counterclockwise around the x-axis
   b) **X-key**: the part turns clockwise around the x-axis
   c) **Y-key**: the part turns counterclockwise around the y-axis
   d) **Y-key**: the part turns clockwise around the y-axis
   e) **z-key**: the part turns counterclockwise around the z-axis
   f) **Z-key**: the part turns clockwise around the z-axis



Mouse:

Rotation with the mouse happens around a fixed point. Keep the **left** mouse button pressed, thus the pointer is projected on the part surface and defines the fixed point. Now move the pointer into the direction of rotation and the part will turn around this fixed point.

17.2.4 Zoom

F3-key:

(1) Switch to the command <Zoom> in the status bar by operating the F3-key.
(2) The part can be zoomed by the arrow keys <up> = zoom out, <down> = zoom in now.

Mouse:

Keep the **right** mouse button pressed and move the pointer up or down.

17.2.5 Pan

F3-key:

(1) Switch to the instruction <Pan> in the status bar by operating the F3-key.
(2) The part can now be shifted by the arrow keys in direction of the arrow or with the x, y, z keys as follows.

x-key: Along the x-axis in positive direction
X-key: Along the x-axis in negative direction
y-key: Along the y-axis in positive direction
Y-key: Along the y-axis in negative direction
z-key: Along the z-axis in positive direction
Z-key: Along the z-axis in negative direction

Mouse:

Keep the **middle** mouse button pressed and move the pointer into the direction, in which you want to move the part.

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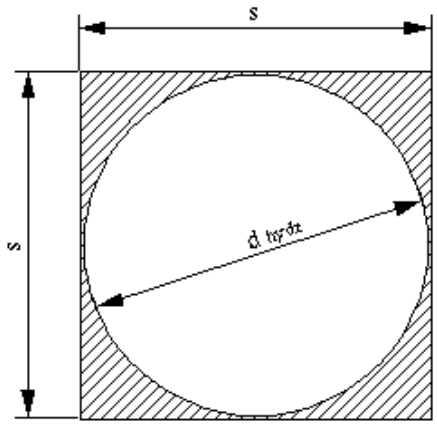
20. Appendix

Hydraulic Diameter

Sketch:



Formula:

Circle cross-section:

$d_{hyd} = 4 * A / U$

$= 4 * \pi d^2 / 4 * \pi d$

$= d$

Square cross-section:

$d_{hyd} = 4 A / U$

$= 4 s^2 / (4 s)$

$= s$

with A = Area of cross section, U = Perimeter in contact with fluid

STL (stereolithography):

Data format for 3D-CAD-Data for rapid prototyping. Closed description of the surface of 3D solids by means of triangular facets. A facet is described by its 3 corner points and its surface normal vector. Semi-standard for rapid prototyping. Abstraction of true surface information. Typical errors are: gaps, overlapping or intersection facets and mis-orientation of normal vectors.

Rapid prototyping:

Procedure for the quick production of prototypes, supported by access to 3D-CAD-Data.

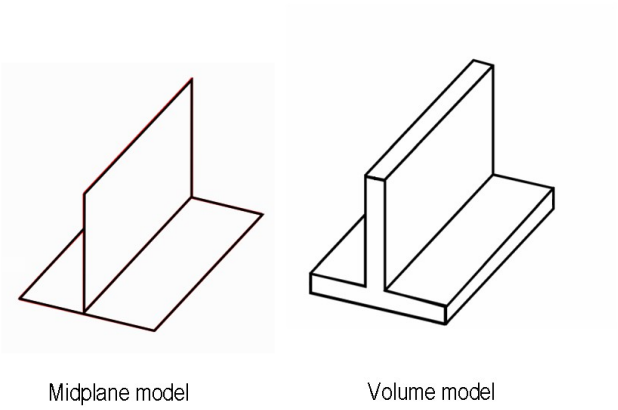
Technical Explanation of the CADMOULD® 3D-F Technology

Dr. Paul F. Filz, Dr. F. Karl Webelhaus

For the calculation of fluid flow within 3 dimensional objects like in cavities of injection moulds for plastics there are two classical Finite Element methods (Fig. 1):

- Midplane model
- Volume model

Midplane Model



The midplane model calculates the flow in the midplane of the geometry. The geometry is represented in a 3 dimensional way but without showing the wall thickness. The wall thickness is numerically assigned to the midplane elements of the geometry as a property.
The midplane model is usable for geometries where the thickness is small compared to the other dimensions. Most injection molded plastic parts are thin walled and fulfill this demand.

Fig. 1: Classical models

Plastic melts flow laminar, i.e. in layers without perpendicular flow, because of their high viscosity. Turbulent flow does not occur.[1] Temperature, shear rate and viscosity, among others, change strongly across the wall thickness. To describe this accurately the midplane model is divided into many layers across the wall thickness.

Advantages of the midplane model are:

- High resolution in areas with substantial changes e.g. of the temperature and the shear rate by dividing the wall thickness into many layers
- easy geometry changing for optimization e.g. flow leader definable independent of CAD in the Finite Element model
- fast calculation
- exact results

1 Reynolds no. normally << 1. Turbulent flow starts at Reynolds no. > 2100. Even in thin gates, areas of high shear rates, Reynolds no. < 20 /Lit. 1/

Disadvantages of the midplane model are:

- the midplane model mostly cannot be created completely automatically from the CAD geometry
- Creating the midplane model means time consuming interactive computer work for the user and needs certain experience /Lit 2/
- The user has to decide which are the flow or thickness directions at areas with large thickness
- Interpretation of the results is more difficult due to the geometrical abstraction (no representation of the wall thickness in the geometrical model)

Volume model

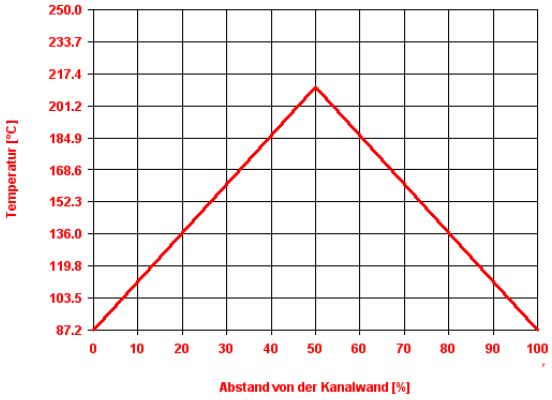
The volume model calculates the flow in all three spatial directions. Here the wall thickness is visible because it is part of the model. Layers are not used.

Advantages of the volume model are:

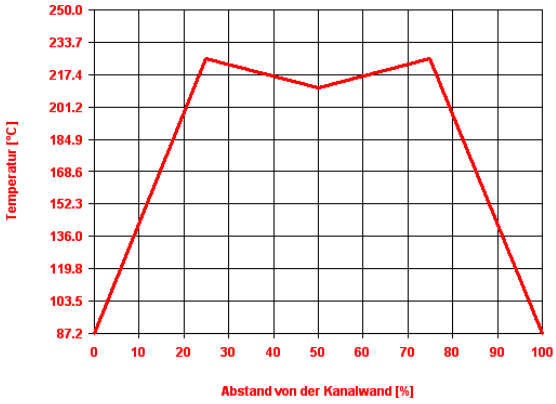
- All spatial directions are equal; the geometry must not be divided into flow or thickness directions by the user.
- By using simple Finite Elements the creation of the geometry can be performed automatically
- Realistic representation of the geometry

Disadvantages of the volume model are:

- Also simple changes in the geometry need tedious interactive work by the user e.g. changing of the design in the CAD system and redoing of the Finite Element mesh generation
- For reasonable accuracy a high number of Finite Elements also across the wall thickness is necessary
- Special user know-how for optimal mesh generation is necessary (e.g. accuracy vs. calculation time)
- Large numbers of elements lead to unacceptable long calculation times /Lit 3/.
- Too coarse discretisation in wall thickness direction leads to inaccurate results (Fig. 2)



3 volume elements across wall thickness



5 volume elements across wall thickness

Fig 2: Temperature distribution across wall thickness with coarse discretisation of the volume model

CADMOULD® 3D-F Model

This was the reason why ***simcon*** developed the patented 3D framework model (3D-F).

The 3D-F Finite Element model is especially adapted for plastic injection molding simulation. It connects the advantages of the two classical models without having their disadvantages.

As basis the 3D-F method uses the geometry description in stl format which was developed for Rapid Prototyping (stereolithography). The stl model is generated automatically by the CAD system. The described part volume is filled additionally by many „tubes“ in the inside, in which fluid flow occurs and by which mechanical forces e.g. for shrinkage and warpage calculation can be transferred. Thus the 3D framework is created, which extends in all three spatial dimensions (x, y, z) (Fig. 3).

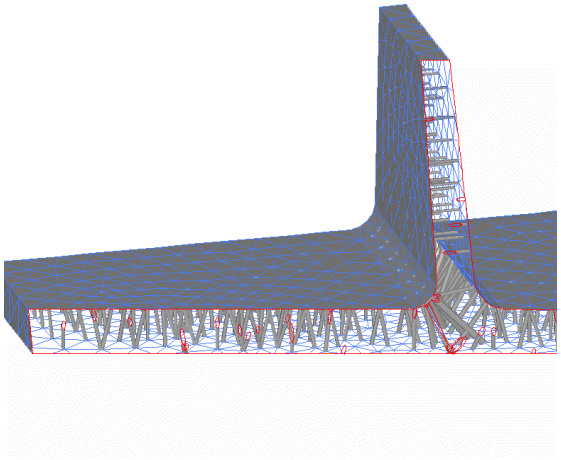


Fig. 3: CADMOULD® 3D-F Framework geometry model[2]

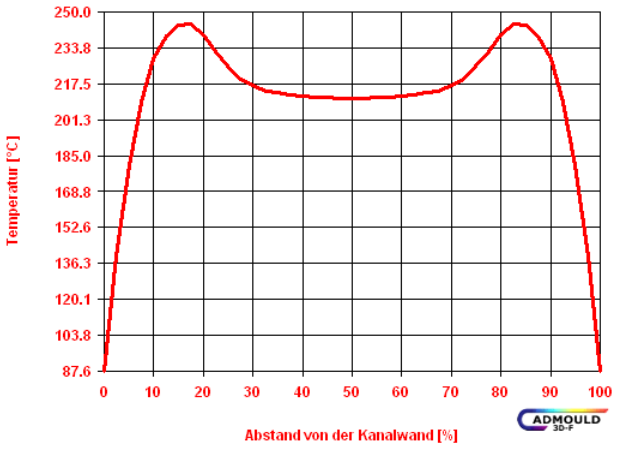


Fig 4: Accurate CADMOULD® 3D-F calculation with high resolution across the part thickness

2Tube diameters are represented smaller than in reality for better representation.

It allows very accurate resolution of the shear rates, temperatures, etc. varying across the wall thickness. (Fig. 4)

This is a necessary prerequisite for accurate calculation of the locally different, complex fluid flow conditions like pressure, velocity, the resulting clamping force and fiber orientations. This and an accurate thermal calculation are also preconditions for accurate determination of shrinkage and warpage.

To achieve the necessary high resolution, in the CADMOULD® 3D-F Model the part is discretized across the wall thickness in 25 sampling points in all areas.

Advantages of the CADMOULD® 3D-F model are:

- Automatic geometry creation
- High resolution in areas of substantial changes e.g. of temperature and shear rate across the wall thickness
- Volumetric part areas and areas with steps in wall thickness are represented and considered accurately
- easy geometry changing for optimization e.g. flow leader definable independent of CAD in the Finite Element model
- Realistic geometry representation
- The feedback of the optimized geometry shape (compensation of shrinkage and warpage) to the CAD system is possible.
- Fast calculation
- Exact results

Conclusion

In addition to save application, CADMOULD® 3D-F with its patented 3D framework model gives the industrial user exactly what he expects from a process simulation software:

- Easy, automatic model creation
- Fast calculation
- Exact results

Authors

Dr. Paul F. Filz is managing director of ***sim**con* kunststofftechnische Software GmbH

Dr. F. Karl Webelhaus is head of development of ***sim**con* kunststofftechnische Software GmbH.

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